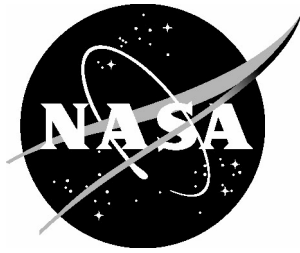


NASA/CP-2004-213028/PT2



COMSAC: Computational Methods for Stability and Control

*Compiled by
C. Michael Fremaux and Robert M. Hall
Langley Research Center, Hampton, Virginia*

April 2004

The NASA STI Program Office . . . in Profile

Since its founding, NASA has been dedicated to the advancement of aeronautics and space science. The NASA Scientific and Technical Information (STI) Program Office plays a key part in helping NASA maintain this important role.

The NASA STI Program Office is operated by Langley Research Center, the lead center for NASA's scientific and technical information. The NASA STI Program Office provides access to the NASA STI Database, the largest collection of aeronautical and space science STI in the world. The Program Office is also NASA's institutional mechanism for disseminating the results of its research and development activities. These results are published by NASA in the NASA STI Report Series, which includes the following report types:

- **TECHNICAL PUBLICATION.** Reports of completed research or a major significant phase of research that present the results of NASA programs and include extensive data or theoretical analysis. Includes compilations of significant scientific and technical data and information deemed to be of continuing reference value. NASA counterpart of peer-reviewed formal professional papers, but having less stringent limitations on manuscript length and extent of graphic presentations.
- **TECHNICAL MEMORANDUM.** Scientific and technical findings that are preliminary or of specialized interest, e.g., quick release reports, working papers, and bibliographies that contain minimal annotation. Does not contain extensive analysis.
- **CONTRACTOR REPORT.** Scientific and technical findings by NASA-sponsored contractors and grantees.

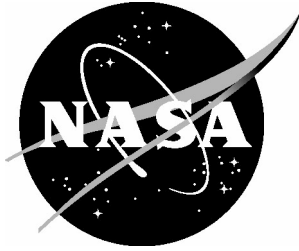
- **CONFERENCE PUBLICATION.** Collected papers from scientific and technical conferences, symposia, seminars, or other meetings sponsored or co-sponsored by NASA.
- **SPECIAL PUBLICATION.** Scientific, technical, or historical information from NASA programs, projects, and missions, often concerned with subjects having substantial public interest.
- **TECHNICAL TRANSLATION.** English-language translations of foreign scientific and technical material pertinent to NASA's mission.

Specialized services that complement the STI Program Office's diverse offerings include creating custom thesauri, building customized databases, organizing and publishing research results ... even providing videos.

For more information about the NASA STI Program Office, see the following:

- Access the NASA STI Program Home Page at [*http://www.sti.nasa.gov*](http://www.sti.nasa.gov)
- E-mail your question via the Internet to [*help@sti.nasa.gov*](mailto:help@sti.nasa.gov)
- Fax your question to the NASA STI Help Desk at (301) 621-0134
- Phone the NASA STI Help Desk at (301) 621-0390
- Write to:
NASA STI Help Desk
NASA Center for AeroSpace Information
7121 Standard Drive
Hanover, MD 21076-1320

NASA/CP-2004-213028/PT2



COMSAC: Computational Methods for Stability and Control

*Compiled by
C. Michael Fremaux and Robert M. Hall
Langley Research Center, Hampton, Virginia*

Proceedings of a symposium sponsored by the National Aeronautics
and Space Administration, Washington, DC, and
held at the Holiday Inn and Conference
Center, Hampton, Virginia
September 23-25, 2003

National Aeronautics and
Space Administration

Langley Research Center
Hampton, Virginia 23681-2199

April 2004

Acknowledgments

The organizing committee acknowledges the highly professional administrative contributions of Ms. Susan N. Price of the NASA Langley Management Support Office. In addition, the committee thanks Mr. Douglas N. Ball (Boeing Commercial Airplanes), Dr. Pradeep Raj (Lockheed Martin), and Mr. John W. Clark, Jr. (NAVAIR) for their participation in the COMSAC planning sessions, and for sharing their personal observations and opinions in the symposium.

The use of trademarks or names of manufacturers in the report is for accurate reporting and does not constitute an official endorsement, either expressed or implied, of such products or manufacturers by the National Aeronautics and Space Administration.
--

Available from:

NASA Center for AeroSpace Information (CASI)
7121 Standard Drive
Hanover, MD 21076-1320
(301) 621-0390

National Technical Information Service (NTIS)
5285 Port Royal Road
Springfield, VA 22161-2171
(703) 605-6000

Executive Summary

A group of nearly 100 technical professionals from government, industry, and academia met in Hampton, Virginia on September 23-25, 2003, for a NASA-sponsored symposium on Computational Methods for Stability and Control (COMSAC) to discuss the status, opportunities, and challenges of applying Computational Fluid Dynamics (CFD) methodology to current and future issues in the field of aircraft stability and control (S&C). The unprecedented advances now being made in CFD technology have demonstrated the powerful capabilities of codes in applications to civil and military vehicles. Used in conjunction with wind-tunnel and flight investigations, many codes are now routinely used by designers in diverse applications such as aerodynamic performance predictions and propulsion integration. Typically, these codes are most reliable for attached, steady, and predominantly turbulent flows. As a result of increasing reliability and confidence in CFD, wind-tunnel testing for some new configurations has been substantially reduced in key areas, such as wing trade studies for mission performance guarantees.

Interest is now growing in the application of CFD methods to other critical design challenges. One of the most important disciplinary elements for civil and military aircraft is S&C. Experience has shown that predictions and analyses of aerodynamic S&C characteristics for full-scale aircraft can be in serious error because of Reynolds number effects, configuration sensitivities, dynamic motion effects, and other issues. Existing experimental facilities may not even be capable of replicating the motions required for aerodynamic measurements. As a result of these shortcomings, a major portion of aircraft development wind-tunnel time (about 60-70%) is typically devoted to S&C testing, especially for various off-design conditions ranging from takeoff and landing to cruise and maneuver. Even with an enormous amount of experimental work, pre-flight aerodynamic prediction errors result in unacceptable increases in program costs, “fly and try” approaches to fixing deficiencies, and extensive developmental delays. Unfortunately, applications of current and emerging CFD codes to engineering analysis in the field of aircraft S&C have been extremely limited. Although isolated examples of success have been demonstrated for certain configurations, the more global issues in S&C – which may involve massive flow separation, unsteady and nonlinear phenomena, dynamic effects, and other extremely complex factors – have not yet been significantly addressed by the CFD community. The current lack of COMSAC-related activities has been further aggravated by the fact that, in contrast to the areas of CFD and performance, very little cross-cultural interaction and communication appears to occur between participants in the areas of CFD and S&C. Within the aerospace community, it is generally agreed that the field of CFD has rapidly matured to the point that the next high payoff applications could occur in S&C. In particular, CFD offers the potential for significantly increasing the basic understanding, prediction, and control of flow phenomena associated with requirements for satisfactory aircraft handling characteristics.

The objectives of the 3-day symposium were to:

1. Discuss the unique aerodynamic phenomena and issues of S&C
2. Define the current characteristics, capabilities, and limitations of CFD codes
3. Define additional or new code requirements for S&C applications

4. Identify potential approaches to develop validated codes
5. Discuss the potential contents and funding opportunities for a COMSAC program

The scope of technical discussions covered civil and military aircraft, including commercial transports, business jets, fighter and attack aircraft, military transports, and bombers. Discussions were limited to fixed-wing aircraft. All sessions were unclassified, and all non-proprietary presentations were collated in the form of PowerPoint presentations with note pages for post-meeting distribution to attendees.

Presentations by speakers described numerous examples of severe impacts of erroneous aerodynamic predictions on the stability and control characteristics of civil and military aircraft. Typically, resolving and mitigating unexpected aerodynamic behavior involved laborious “cut and fly” approaches required during critical flight test programs. These shortcomings resulted in significant program delays, costs, mission limitations, non-optimum configurations (weight, capabilities, etc.), and severe scrutiny by stakeholders and customers.

In-depth discussions of specific experiences with actual applications of various levels of computational methods to S&C indicated a wide range of success and an overriding sense of skepticism by the attendees. After individual presentations were made to provide organizational and individual perspectives on CFD for S&C, the attendees were briefed on NASA’s vision of a COMSAC program. Comments were solicited to identify and prioritize technology areas for such a program. Finally, the Director of the NASA-Langley Aerospace Vehicle Systems Technology Office shared his view of a potential strategy to augment funding and program priority in this area.

The general findings of the workshop were:

1. Inaccurate prediction of aerodynamic stability and control parameters continues to have major cost and programmatic impacts in virtually every vehicle class. These impacts include unacceptable increases in program costs, “fly and try” approaches to fixing deficiencies, extensive developmental delays and profit losses due to delayed deliveries.
2. Prediction of the character of separated flows across the speed range (with the attendant issues of transition prediction, turbulence modeling, unsteady flows, etc.) and the impact of separated flow on aircraft S&C should receive priority in a COMSAC program.
3. A pervasive attitude of skepticism regarding the success of CFD applications to aircraft S&C issues (especially for preliminary and conceptual design) exists within the CFD community, as well as the S&C community.
4. The application of advanced and emerging CFD methods as design tools will be dependent on the accumulation and demonstrated success of experiences for both generic and specific aircraft configurations.

5. Issues regarding the CFD process (cost, time required, adaptive gridding requirements, error quantification, etc.) should be high priority targets for COMSAC efforts.
6. One of the most valuable contributions of the symposium was the mechanism to share perspectives and experiences between the diverse CFD specialists and S&C specialists. Prior to this meeting, communication between these two groups was extremely poor, resulting in a major barrier to the acceleration and acceptance of CFD methods for S&C applications.

Joseph R. Chambers
ViGYAN, Inc.
Hampton, Virginia

Contents

Executive Summary	iii
Attendees	x

Part 1

Introductory Remarks	1
Darrel R. Tenney Aerospace Vehicle Systems Technology Office, NASA Langley Research Center, Hampton, Virginia	
Introduction to Computational Methods for Stability and Control (COMSAC)	7
Robert M. Hall and C. Michael Fremaux NASA Langley Research Center, Hampton, Virginia Joseph R. Chambers ViGYAN, Inc., Hampton, Virginia	
Stability & Control Challenges for COMSAC: a NASA Langley Perspective	28
C. Michael Fremaux NASA Langley Research Center, Hampton, Virginia	
Emerging CFD Capabilities and Outlook – A NASA Langley Perspective	48
Robert T. Biedron, S. Paul Pao, and James L. Thomas NASA Langley Research Center, Hampton, Virginia	
The Role for Computational Fluid Dynamics for Stability and Control – Is it Time?	69
Douglas N. Ball Boeing Commercial Airplanes, Renton, Washington	
Northrop Grumman Perspective on COMSAC	98
Dale Lorincz Northrop Grumman Air Combat Systems, El Segundo, California	
Boeing Integrated Defense Systems Perspective on COMSAC	118
David M. Evans Boeing Integrated Defense Systems – Tactical Aircraft and Weapons, St. Louis, Missouri	
Computational Methods in Stability and Control – WPAFB Perspective	124
William Blake AFRL Air Vehicles Directorate, Wright Patterson Air Force Base, Ohio William Thomas Aeronautical Systems Center, Wright Patterson Air Force Base, Ohio	

Perspective: Raytheon Aircraft Company.....	144
Neal J. Pfeiffer and Dana C. Herring	
Raytheon Aircraft Company, Wichita, Kansas	
A Greybeard’s View of the State of Aerodynamic Prediction.....	181
Tom Lawrence	
Aeromechanics Division, Naval Air Systems Command, Patuxent River, Maryland	
Computational Methods for Stability and Control: A Perspective.....	214
Pradeep Raj	
Lockheed Martin Aeronautics Company, Marietta, Georgia	
Boeing TacAir Stability and Control Issues for Computational Fluid Dynamics.....	227
William B. Hollingsworth	
Boeing TacAir, St. Louis, Missouri	
NAVAIR S&C Issues for CFD.....	239
Steve Donaldson	
Flight Dynamics Branch, Naval Air Systems Command, Patuxent River, Maryland	
An S&C Perspective on CFD.....	248
Russ D. Killingsworth	
JSF Stability and Control, Lockheed Martin Aeronautics, Fort Worth, Texas	
Issues, Challenges & Payoffs: A Boeing User’s Perspective on CFD for S&C.....	273
D. R. Bogue, T. R. Lines, and R. D. Doll	
Boeing Commercial Airplanes, Seattle, Washington	
Stability and Control in Computational Simulations for Conceptual and Preliminary Design: the Past, Today, and Future?.....	309
William H. Mason	
Department of Aerospace and Ocean Engineering, Virginia Tech, Blacksburg, Virginia	
Computational Methods for Dynamic S&C Derivatives.....	341
Lawrence L. Green and Patrick C. Murphy	
NASA Langley Research Center, Hampton, Virginia	
Angela M. Spence	
Department of Aerospace Engineering, Mississippi State University, Starkville, Mississippi	

Part 2

Boeing TacAir CFD Capabilities/Issues.....	365
David Stookesberry and Frank C. Berrier	
Boeing TacAir, St. Louis, Missouri	

TetrUSS Capabilities for S&C Applications.....	378
Neal T. Frink and Paresh C. Parikh NASA Langley Research Center, Hampton, Virginia	
Computational Simulations for Stability and Control – BCA State-of-the-Art	396
N. J. Yu, T. J. Kao, D. R. Bogue, and T. R. Lines Boeing Commercial Airplanes, Seattle, Washington	
Time-Accurate Computational Simulation.....	417
S. Paul Pao and Pieter G. Buning NASA Langley Research Center, Hampton, Virginia	
Application of CFD to Abrupt Wing Stall Using RANS and DES.....	433
James R. Forsythe Cobalt Solutions, LLC, Springfield, Ohio	
Application of Computational Stability and Control Techniques Including Unsteady Aerodynamics and Aeroelastic Effects.....	489
David M. Schuster and John W. Edwards NASA Langley Research Center, Hampton, Virginia	
Hinge Moment Predictions Using CFD.....	511
M. J. Grismer, D. Kinsey, and D. Grismer Air Vehicles Directorate, Air Force Research Laboratory, Wright Patterson Air Force Base, Ohio	
CFD Simulation of Aircraft in Coning Motion.....	533
Syta Saephan and C. P. van Dam Department of Mechanical and Aeronautical Engineering, University of California, Davis, California	
Use of CFD-Generated Aerodynamic Data for an F-15E/SLV Flying/Handling Qualities Analysis.....	565
Jeffery A. Batte and W. Shawn Westmoreland Jacobs Sverdrup, Eglin Air Force Base, Florida	
Rapid Euler CFD for High-Performance Aircraft Design.....	593
Eric F. Charlton Lockheed Martin Aeronautics Company, Fort Worth, Texas	
Quantitative Prediction of Computational Quality.....	638
Michael J. Hemsch, James M. Luckring, and Joseph H. Morrison NASA Langley Research Center, Hampton, Virginia	
Best Practices System to Enhance CFD Use in Stability and Control Applications.....	656
Michael R. Mendenhall Nielsen Engineering & Research, Inc., Mountain View, California	

Dynamic Water Tunnel Testing for Code Benchmarking.....	671
Brooke C. Smith and John Hodgkinson	
AeroArts LLC, Palos Verdes Peninsula, California	
COMSAC: Vision and Potential Program Planning.....	692
Robert M. Hall and C. Michael Fremaux	
NASA Langley Research Center, Hampton, Virginia	
Joseph R. Chambers	
ViGYAN, Inc., Hampton, Virginia	
COMSAC Feedback.....	718
Pradeep Raj	
Lockheed Martin Aeronautics Company, Marietta, Georgia	
COMSAC Plan: NAVAIR Comments.....	722
John W. Clark	
Naval Air Systems Command, Patuxent River, Maryland	
COMSAC Feedback – Boeing Commercial Airplanes.....	730
Douglas N. Ball	
Boeing Commercial Airplanes, Renton, Washington	

NASA Langley Symposium on Computational Methods for Stability and Control

Hampton Holiday Inn and Conference Center
Hampton, Virginia
September 23 – 25, 2003

Attendee List

1. Anderson, Dr. William K.
UT SimCenter at Chattanooga
Two Union Square, Suite 300
Chattanooga, TN 37402
kyle-anderson@utc.edu
423-648-0343
2. Arabshahi, Dr. Abdollah
UT SimCenter at Chattanooga
Two Union Square, Suite 300
Chattanooga, TN 37402
Abi-Arabshahi@utc.edu
423-648-0342
3. Ashbaugh, Mr. William H.
ASC/AAAV
2145 Monahan Way, Bldg 28
Wright-Patterson AFB, OH 45433
william.ashbaugh@wpafb.af.mil
937-255-7210,x-3864
4. Ball, Mr. Douglas N.
Boeing Commercial Airplanes
535 Garden Avenue N., Mail Stop
67-LH Renton, WA 98055
douglas.n.ball@boeing.com
425-234-1016
5. Batte, Mr. Jeffery A.
Jacobs Sverdrup
308 West D Ave, Suite 1
P.O. Box 1935
Eglin AFB, FL 32542
batte@eglin.af.mil
850-882-0398
6. Berrier, Mr. Frank C.
Boeing TacAir
PO Box 516, MC S1066450
St. Louis, MO 63166
frank.berrier@boeing.com
314-232-5774
7. Biedron, Dr. Robert T.
M.S. 128
NASA Langley Research Center
Hampton, VA 23681
robert.t.biedron@nasa.gov
757-864-2156
8. Blake, Mr. William
Air Force Research Laboratory
AFRL/VACA
WPAFB, OH 45433
william.blake2@wpafb.af.mil
937-255-6764
9. Bogue, Mr. David
Boeing Commercial Airplanes
15530 Bothell Way NE #111
Seattle, WA 98155
david.r.bogue@boeing.com
425-234-1078
10. Bower, Dr. Daniel R.
National Transportation Safety
Board
490 L'Enfant Plaza East
Washington, DC 20594
bowerd@ntsb.gov
202-314-6562

11. Chambers, Mr. Joseph R.
ViGYAN, Inc.
205 Old Dominion Rd.
Yorktown, VA 23692
jrchambers@cox.net
757-898-6080
12. Chung, Dr. James J.
NAVAIR
Bldg. 2187, Suite 1320B, Unit5
48110 Shaw Road
Patuxent River, MD 20670
chungjj@navair.navy.mil
301-342-8547
13. Clark, Jr., Mr. John W.
NAVAIR
3040 Blackberry Ln.
Prince Frederick, MD 20678
clarkjw1@navair.navy.mil
301-342-8550
14. Craft, Mr. John W.
Boeing, Huntington Beach
5301 Bolsa Ave., MC H013-B318
Huntington Beach, CA 92647
john.w.craft@boeing.com
714-235-8261
15. Crider, Mr. Dennis A.
National Transportation Safety
Board
490 L'Enfant Plaza East, SW
RE-60
Washington, D.C. 20594
criderd@ntsb.gov
202-314-6564
16. Donaldson, Mr. Steve A.
NAVAIR
B2187, Suite 1390A
48110 Shaw Rd., Unit 5
Patuxent River, MD 20670
donaldsonsa@navair.navy.mil
301-342-0282
17. Dreyer, Dr. James J.
Applied Research Laboratory
Penn State University
P.O. Box 30
State College, PA 16804
jjd@wt.arl.psu.edu
814-863-3018
18. Edwards, Dr. John W.
MS 340
NASA Langley Research Center
Hampton, VA 23681
john.w.edwards@nasa.gov
757-864-2273
19. Evans, Mr. David M.
Boeing
Box 516, MS 2703760
St. Louis, MO 63166
david.m.evans@boeing.com
314-233-6290
20. Forsythe, Dr. James R.
Cobalt Solutions, LLC
4636 New Carlisle Pike
Springfield, OH 45504
forsythe@cobaltcfd.com
937-620-5938
21. Fremaux, Mr. Charles M.
MS 153
NASA Langley Research Center
Hampton, VA 23681
charles.m.fremaux@nasa.gov
757-864-1193
22. Frink, Dr. Neal T.
MS 499
NASA Langley Research Center
Hampton, VA 23681
neal.t.frink@nasa.gov
757-864-2864

23. Garb, Mr. Slava Z.
Gulfstream Aerospace
PO Box 2206, M/S D-04
Savannah, GA 31402
slava.garb@gulfaero.com
912-965-3527
24. Garcia, Mr. Garrett M.
U.S. Air Force
75 Vandenburg Drive, Bldg 1630
ESC/MAV
Hanscom AFB, MA 01731
garrett.garcia@hanscom.af.mil
719-964-7377
25. Ghaffari, Mr. Farhad
MS 286
NASA Langley Research Center
Hampton, VA 23681
Farhad.Ghaffari-1@nasa.gov
757-864-2856
26. Goble, Dr. Brian D.
Lockheed Martin Aeronautics Co.
101 Academy Blvd., Mail Zone 8656
P.O. Box 748
Fort Worth, TX 76101
brian.d.goble@lmco.com
817-935-4707
27. Gopalarathnam, Dr. Ashok
North Carolina State University
Mech. and Aerospace Engineering
Box 7910
Raleigh, NC 27695
ashok_g@ncsu.edu
919-515-5669
28. Green, Mr. Lawrence L.
MS 159
NASA Langley Research Center
Hampton, VA 23681
Lawrence.L.Green@nasa.gov
757-864-2228
29. Grismer, Dr. Matthew J.
AFRL/VAAC, B146 R225
2210 Eighth Street
Wright-Patterson AFB, OH 45433
matthew.grismer@wpafb.af.mil
937-255-3876
30. Grove, Mr. Darren V.
NAVAIR,
Bldg. 2187, Suite 1320-D4
48110 Shaw Rd.
Patuxent River, MD 20670
grovedv@navair.navy.mil
301-342-8562
31. Guruswamy, Dr. Guru
T27B-1
NASA Ames Research Center
Moffett Field, CA 94035
Guru.p.guruswamy@nasa.gov
650-604-6329
32. Hall, Dr. Robert M.
MS 499
NASA Langley Research Center
Hampton, VA 23681
robert.m.hall@nasa.gov
757-864-2883
33. Heim, Mr. Eugene H.
MS 153
NASA Langley Research Center
Hampton, VA 23681
eugene.h.heim@nasa.gov
757-864-9638
34. Hensch, Dr. Michael J.
MS 286
NASA Langley Research Center
Hampton, VA 23681
michael.j.hensch@nasa.gov
757-864-2882

35. Herring, Mr. Dana C.
Raytheon Aircraft Company
MS 971-B6, PO Box 85
9709 E Central
Wichita, KS 67201
dana_herring@rac.ray.com
316-676-6718
36. Hickey, Mr. Herbert J. (Skip)
WJN Consulting
5137 Croftshire Drive
Kettering, OH 45440
skiphickey@aol.com
937-433-5391
37. Hollingsworth, Mr. William B.
Boeing TacAir
PO Box 516, MC S1066450
St. Louis, MO 63166
william.b.hollingsworth-iii@boeing.com
314-234-2221
38. Hoyle, Mr. David L.
Lockheed Martin
M/C 0685
86 S. Cobb Drive
Marietta, GA 30063
david.l.hoyle@lmco.com
770-494-8279
39. Jiang, Dr. Minyee J.
U.S. Navy, NSWCCD
9500 MacArthur Blvd.
West Bethesda, MD 20817
jiangm@nswccd.navy.mil
301-227-6090
40. Karman, Dr. Steve L.
UT SimCenter at Chattanooga
Two Union Square, Suite 300
Chattanooga, TN 37402
Steve-Karman@utc.edu
423-648-0595
41. Kerho, Dr. Michael F.
Rolling Hills Research Corporation
3425 Lomita Blvd.
Torrance, CA 90505
MKerho@RollingHillsResearch.com
310-257-9578
42. Killingsworth, Mr. Russ D.
Lockheed Martin Aeronautics
PO Box 748, MZ 6468
Fort Worth, TX 76101
russ.d.killingsworth@lmco.com
817-7632915
43. Kokolios, Mr. Alex
NAVAIR
B2187, Suite 1390A
48110 Shaw Rd., Unit 5
Patuxent River, MD 20670
kokoliosa@navair.navy.mil
301-342-8574
44. Kolly, Dr. Joseph
National Transportation Safety
Board
490 L'Enfant Plaza East, SW
Washington, D.C. 20594
kollyj@ntsb.gov
202-3146622
45. Komerath, Dr. Narayanan M.
Georgia Institute of Technology
School of Aerospace Engineering
Atlanta, GA 30332
narayanan.komerath@ae.gatech.edu
404-894-3017
46. Kramer, Mr. Brian R.
Rolling Hills Research Corporation
3425 Lomita Blvd.
Torrance, CA 90505
BKramer@RollingHillsResearch.com
310-257-9578

47. Kumar, Dr. Ajay
MS 285
NASA Langley Research Center
Hampton, VA 23681
Ajay.Kumar-1@nasa.gov
757-864-3520
48. Laiosa, Mr. Joseph P.
Naval Air Warfare Center
Aircraft Division
Bldg 2187, Suite 1320-E2
48110 Shaw Road, Unit #5
Patuxent River, MD 20670
Laiosajp@navair.navy.mil
301-342-5723
49. Lawrence, Mr. Tom
NAVAIR
Aeromechanics Division
Patuxent River, MD 20670
lawrencejt@navair.navy.mil
301-342-8551
50. Leavitt, Mr. Laurence D.
MS 499
NASA Langley Research Center
Hampton, VA 23681
Laurence.D.Leavitt@nasa.gov
757-864-3017
51. Lee-Rausch, Ms. Elizabeth M.
MS 128
NASA Langley Research Center
Hampton, VA 23681
Elizabeth.M.Lee-Rausch@nasa.gov
757-864-8422
52. Lorincz, Mr. Dale
Northrop Grumman Corp.
One Hornet Way
9L30/W6
El Segundo, CA 90245
dale.lorincz@ngc.com
310-332-9966
53. Luckring, Dr. James M.
MS 286
NASA Langley Research Center
Hampton, VA 23681
James.M.Luckring@nasa.gov
757-864-2869
54. Malone, Dr. John B.
MS 285
NASA Langley Research Center
Hampton, VA 23681
John.B.Malone@nasa.gov
757-864-8988
55. Mangalam, Dr. Siva M.
Tao Systems
471 McLaws Circle
Williamsburg, VA 23185
siva@taosystem.com
757-220-5040
56. Mason, Dr. William H.
Virginia Tech
215 Randolph Hall, MC 0203
Blacksburg, VA 24061
whmason@vt.edu
540-231-6740
57. Masters, Mr. James
Jacobs Sverdrup, AEDC Group
Arnold AFB, TN 37389
james.masters@arnold.af.mil
931-454-7739
58. McNamara, Mr. William G.
NAVAIR/Naval Air Warfare Center
Bldg 2035
48183 Switzer Road
Patuxent River, MD 20670
mcnamaraWG@navair.navy.mil
301-757-0848

59. Mehrotra, Dr. Sudhir C.
ViGYAN, Inc.
30 Research Drive
Hampton, VA 23666
mehrotra@vigyan.com
757-865-1400
60. Mendenhall, Mr. Michael R.
Nielsen Engineering & Research
605 Ellis St., Suite 200
Mountain View, CA 94043
mrm@nearinc.com
650-968-9457
61. Morrison, Mr. Joseph H.
MS 128
NASA Langley Research Center
Hampton, VA 23681
joseph.h.morrison@nasa.gov
757-864-2294
62. Murphy, Dr. Patrick C.
MS 132
NASA Langley Research Center
Hampton, VA 23681
Patrick.C.Murphy@nasa.gov
757-864-4071
63. Murri, Mr. Daniel G.
MS 153
NASA Langley Research Center
Hampton, VA 23681
Daniel.G.Murri@nasa.gov
757-864-1160
64. Nelson, Dr. Robert C.
University of Notre Dame
Department of Aerospace and
Mechanical Engineering
Notre Dame, IN 46556
robert.c.nelson.1@nd.edu
574-631-4733
65. O'Callaghan, Mr. John
National Transportation Safety
Board
490 L'Enfant Plaza E., S.W.
Washington, D.C. 20594
ocallaj@ntsb.gov
202-314-6560
66. Om, Dr. Deepak
Boeing Commercial Airplanes
Mail Code 67-LF, P. O. Box 3707
Seattle, WA 98124
deepak.om@boeing.com
425-234-1116
67. Ozoroski, Ms. Lori P.
MS 348
NASA Langley Research Center
Hampton, VA 23681
Lori.P.Ozoroski@nasa.gov
757-8645992
68. Pao, Dr. S. Paul
MS 499
NASA Langley Research Center
Hampton, VA 23681
s.p.pao@nasa.gov
757-864-3044
69. Parikh, Dr. Paresh C.
MS 499
NASA Langley Research Center
Hampton, VA 23681
paresh.c.parikh@nasa.gov
757-864-2244
70. Park, Mr. Michael A.
MS 128
NASA Langley Research Center
Hampton, VA 23681
Michael.A.Park@nasa.gov
757-864-6604

71. Pfeiffer, Dr. Neal J.
Raytheon Aircraft Company
MS 999-B12 , PO Box 85
9709 E. Central
Wichita, KS 67201
neal_pfeiffer@rac.ray.com
316-676-6775
72. Pirzadeh, Dr. Shahyar Z.
MS 499
NASA Langley Research Center
Hampton, VA 23681
Shahyar.Z.Pirzadeh@nasa.gov
757-864-2245
73. Player, Ms. Jennifer
Bihrl Applied Research, Inc.
18 Research Drive
Hampton, VA 23666
jplayer@bihrl.com
757-766-2416
74. Raj, Dr. Pradeep
Lockheed Martin Aeronautics Co.
86 S. Cobb Dr.
D/1AEM, Z/0660
Marietta, GA 30063
pradeep.raj@lmco.com
770-494-3801
75. Ralston, Mr. John
Bihrl Applied Research, Inc.
18 Research Drive
Hampton, VA 23666
jralston@bihrl.com
757-766-2416
76. Ratvasky, Mr. Thomas P.
21000 Brookpark Rd, Mail Stop 11-2
NASA Glenn Research Center
Cleveland, OH 44135
thomas.p.ratvasky@nasa.gov
216-433-3905
77. Roberts, Mr. Michael
University of California, Davis
Dept. of Mechanical & Aeronautical
Engineering
1 Shields Ave.,
Davis, CA 95616
mike_m_roberts@yahoo.com
530-752-2261
78. Saephan, Mr. Syta
University of California, Davis
Dept. of Mechanical & Aeronautical
Engineering
1 Shields Ave.,
Davis, CA 95616
ssaephan@ucdavis.edu
530-752-2261
79. Schuster, Dr. David M.
MS 340
NASA Langley Research Center
Hampton, VA 23681
david.m.schuster@nasa.gov
757-864-2259
80. Shah, Mr. Gautam H.
MS 153
NASA Langley Research Center
Hampton, VA 23681
gautam.h.shah@nasa.gov
757-864-1163
81. Skujins, Dr. Ojars
Aeronautical Systems Center
ASC/ENFT
2530 Loop Road West
Wright-Patterson AFB, OH 45433
ojars.skujins@wpafb.af.mil
937-255-4166
82. Slomski, Dr. Joe
NSWC Carderock Division
9500 MacArthur Boulevard
West Bethesda, MD 20817
slomskijf@nswccd.navy.mil
301-227-1935

83. Smith, Mr. Brooke C.
AeroArts LLC
P.O. Box 2909
Palos Verde Peninsula, CA 90274
Brooke.Smith@AeroArts.com
310-547-0927
84. Stookesberry, Mr. David C.
Boeing
P.O. Box 516, MC S106-6420
St. Louis, MO 63166
david.c.stookesberry@boeing.com
314-233-6346
85. Stuckert, Dr. Gregory K.
Fluent Inc.
10 Cavendish Court
Lebanon, NH 03766
gks@fluent.com
603-643-2600, x243
86. Stuever, Mr. Robert A.
The Boeing Company
Wichita Development &
Modification Center
P. O. Box 7730, MC K05-14
Wichita, KS 67277
Robert.A.Stuever@boeing.com
316-523-4826
87. Sung, Dr. Chao Ho
Dept of the Navy
9500 MacArthur Blvd
West Bethesda, MD 20817
sungch@nswccd.navy.mil
301-227-1865
88. Tello, Mr. Richard J.
The MITRE Corporation
75 Vandenberg Drive
Attn: Joint Stars
Hanscom AFB, MA 01731
rtello@hanscom.af.mil
781-377-9919
89. Tenney, Dr. Darrel R.
MS 208
NASA Langley Research Center
Hampton, VA 23681
Darrel.R.Tenney@nasa.gov
757-864-6033
90. Thacker, Mr. Michael
Cessna Aircraft Company
5701 E. Pawnee, D363P
Wichita, KS 67218
mthacker@cessna.textron.com
316-831-2807
91. Thomas, Dr. James L.
MS 128
NASA Langley Research Center
Hampton, VA 23681
James.L.Thomas@nasa.gov
757-8645578
92. Van Dam, Dr. Case
University of California, Davis
Dept. of Mechanical & Aeronautical
Engineering
1 Shields Ave.,
Davis, CA 95616
cpvandam@ucdavis.edu
530-752-7741
93. Vassberg, Dr. John C.
The Boeing Company
5301 Bolsa Avenue
MC H013-B318
Huntington Beach, CA 92647
john.c.vassberg@boeing.com
714-896-1607
94. Wahls, Dr. Richard A.
MS 499
NASA Langley Research Center
Hampton, VA 23681
richard.a.wahls@nasa.gov
757-864-5108

95. Westmoreland, Mr. William S.
Jacobs Sverdrup
205 West D Ave.
Eglin AFB, FL 32542
westmore@eglin.af.mil
850-8820918

96. Wong, Dr. Tin-Chee
U. S. Army, AMSAM-RD-AE-A
Bldg 4488
Redstone Arsenal, AL 35898
TinChee.Wong@us.army.mil
256-705-9605

97. Yang, Dr. Cheng I.
Code 5400, Bldg 16
Naval Surface Warfare Center
Bethesda, MD 20084
yangci@nswccd.navy.mil
301-227-4658

98. Yip, Mr. Long P.
MS 254
NASA Langley Research Center
Hampton, VA 23681
Long.P.Yip@nasa.gov
757-864-1700

99. Yu, Dr. Neng J.
Boeing Commercial Airplanes
Enabling Technology & Research
M/S 67-LF, P.O.Box 3707
Seattle, WA 98124
neng.j.yu@pss.boeing.com
425-234-1192



Boeing TacAir CFD Capabilities/Issues

NASA COMSAC Symposium



**Hampton, VA
September 23-25, 2003**



Dave Stookesberry / Frank Berrier



Presentation Perspective

- Tactical Aircraft (TacAir) Group of Boeing's Integrated Defense Systems (IDS) Division
- Production Tactical Fighter Aircraft
 - Primarily aircraft upgrade or manufacturing issues
- Located in St. Louis, Missouri along with elements of Boeing Phantom Works R&D Unit (Advanced Military Aircraft)
- WIND Flow Solver (Steady-state, Navier-Stokes)
 - Two-equation turbulence model
 - Primarily structured/overlapped grids, some unstructured grids
 - CFD solutions run on LINUX PC clusters



CFD Usage for Stability and Control (S&C) Analyses

- Steady-state, Navier-Stokes CFD solutions were computed
 - Two-equation turbulence model
 - Structured/overlapped grids
- CFD used to quantify the aerodynamic impact of outer moldline changes
 - Sensitivity analyses and trade studies
 - Risk reduction for planned wind tunnel or flight tests
- CFD proven useful in understanding flight test anomalies
 - Abruptly occurring events
 - In-flight system failures and mishaps
- CFD has had virtually no predictive or corrective role in forming maneuvering simulation databases.



Boeing TacAir CFD S&C Applications

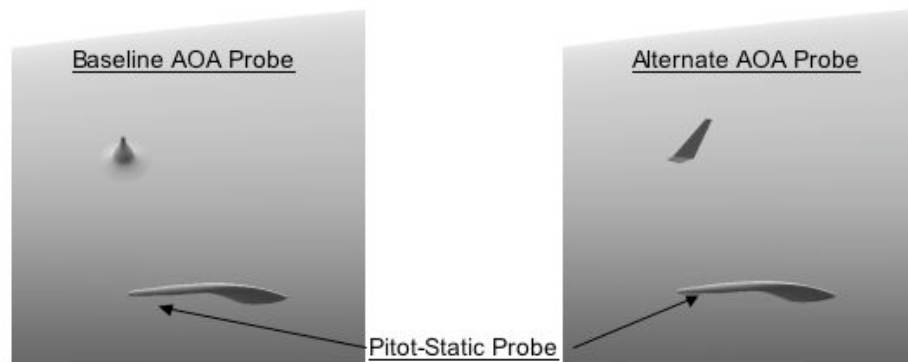


Applications of CFD to stability and control analyses for tactical fighter aircraft:

- Effect of probes, antennae and sensor additions/changes on source error correction terms for flight control laws.
- Effect of lifting/control surface moldline changes on stability.
- Diagnostic of a lateral stability anomaly caused by an asymmetric store loading.
- Effect of large protuberances on stability.
- Characterization of the flowfield and possible causes of abrupt wing stall.

CFD Application # 1

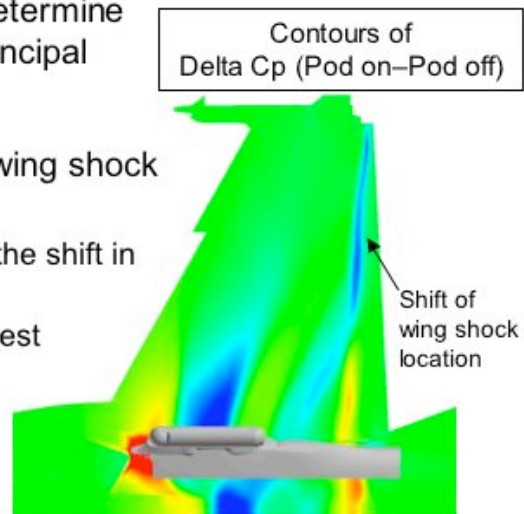
- Background: An alternate angle-of-attack (AOA) probe was proposed as a possible cost reduction initiative.
- CFD Study: Analyze local flowfield effect of changing AOA probe to determine whether planned flight tests would be sufficient (risk reduction).
- Results: CFD analysis showed that changing the AOA probe could have relatively large effect on pitot-static readings.
 - High probability that additional source error correction flights would be needed (thereby increasing cost)



NASA COMSAC Symposium

CFD Application # 2

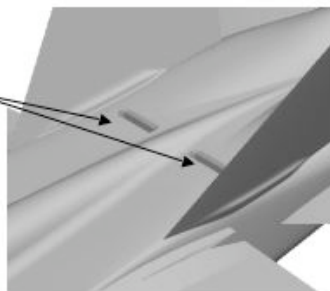
- Background: During flight test, an asymmetric configuration showed an unexpectedly large rolling moment at a specific flight condition.
- CFD Study: Analyze configuration to verify suspected cause of rolling moment and determine whether a specific pod feature was the principal culprit.
- Results: CFD analysis showed a shift in wing shock position caused the rolling moment.
 - Nothing inherent in the pod shape caused the shift in shock position
 - Asymmetric aerodynamic loading was biggest contributor



CFD Application #3

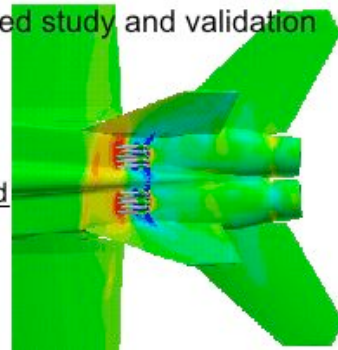
- Background: Preliminary flight tests showed a localized impact on lateral stability due to a change in ECS exhaust configuration.
- CFD Study: Determine if CFD could replicate the impact on lateral stability and if so, determine the contributing factors of the configuration.
- Results: CFD analysis showed negligible effect of the ECS exhaust configuration on lateral stability. Could not replicate flight test results.
 - CFD coefficients for baseline configuration agree well with wind tunnel results
 - Possible configuration and aft end modeling issues need study and validation

ECS Exhaust
Location



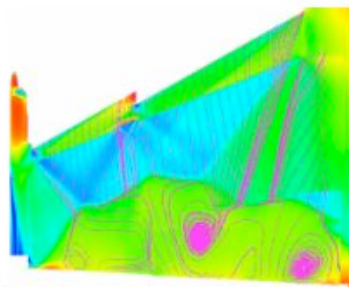
NASA COMSAC Symposium

Contours of
Delta Cp (ECS Mod
– Baseline)



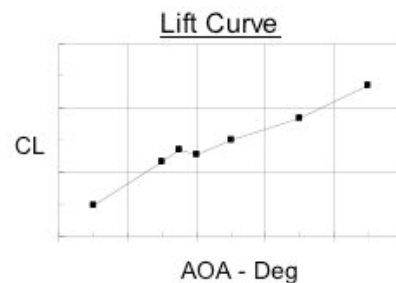
CFD Application #4

- Background: Initial F/A-18E/F flight testing revealed uncommanded rolling motion while maneuvering at transonic flight. Asymmetric flow separation on the wing upper surface.
- CFD Study: Determine whether CFD could predict the lift curve break seen in wind tunnel tests. Obtain a detailed flowfield description of the phenomena.
- Results: Results showed that CFD could help identify characteristics associated with causing abrupt wing stall.



NASA COMSAC Symposium

Contours of
Surface Pressure and
Particle Traces





CFD Application #5

- Background: A series of flight tests were planned to optimize a moldline change in order to reduce the performance / manufacturing impact. Wind tunnel testing is not sufficient to determine if an optimized moldline change would still meet the S&C requirements.
- CFD Study: Use CFD to screen moldline concepts and perhaps shorten the flight test.
- Results: Compared CFD computed flowfields with the unoptimized-moldline flowfield to determine the most promising versions to meet S&C requirements. The flight test sequence was revised to test the most likely versions earlier thereby reducing the number of flights needed.



Summary of Boeing TacAir CFD Application Results



- In all cases, steady state CFD results were generated.
- CFD results were, at a minimum, qualitatively useful.
- Quantitatively useful results were obtained in certain cases.
 - Analyses of minor moldline changes
 - Analysis of influences on the local flowfield
- CFD used extensively for abrupt wing stall analyses,
 - Results gave a qualitative assessment of overall flowfield changes
 - Amount of unsteadiness could not be quantified
 - Correlation of CFD, wind tunnel and flight test results led to better understanding of the rapidly changing flowfield



General Obstacles to CFD Use

- Time required to prepare a sufficiently robust and accurate grid for complex geometry often precludes CFD use.
- Need for large CPU capacity and highly specialized personnel can place elements of a business in competition for the same resources.
- Run time to obtain CFD results, especially for unsteady analyses, compound the CPU resource problem.
- Lack of validation for CFD solutions involved in stability and control analyses is a major drawback for more CFD use.



Specific Impediments to Boeing TacAir S&C CFD

Characteristics of TacAir S&C Prediction and Testing

- Maneuvering Flight (Changing flight conditions)
- Complicated/Changing Geometry (Control surfaces, gaps, airframe flexibility, stores)
- Unsteady/Complex Flow (Laminar/turbulent separation, shock interactions)
- Flow Control Devices (Fences, slats, porosity, vortex generators)
- Flow Anomalies Frequently Occur over Narrow Range of Flight Conditions
- Limited Time to Fix/Understand Problem (Usually in flight test or airplane delivery phase)

Impediments to Application of CFD

- Solution speed, CPU availability to get multiple solutions.
- Geometry acquisition / grid generation time. Robustness/accuracy of complicated grids.
- Time-dependent solution accuracy/cost. Turbulence/transition model accuracy/cost.
- Geometric complexity. Accuracy of boundary condition models.
- Difficult to predict flow anomaly with limited number of CFD runs.
- Set-up time, solution speed, CPU availability, prior validation.

Validation of CFD is needed to show that extra time/cost for CFD is worthwhile.



Recommendations for COMSAC

Recommendations for improving/increasing use of CFD for Tactical Fighter Aircraft S&C analyses:

- Increase validation to quantify the current capability of S&C CFD analysis
- Improve geometry acquisition and grid generation processes
 - Reduce the time to create quality complex-geometry grids.
- Improve CFD flow solvers to reduce the solution time/cost
 - Necessary for unsteady time-dependent analyses.
- Improve the accuracy and robustness of boundary layer models (i.e., transition, porosity, flow control jets)
 - Flow control devices are becoming more prevalent in newer vehicles.

NASA COMSAC Symposium



TetrUSS Capabilities for S&C Applications

Neal T. Frink and Paresh Parikh
NASA Langley Research Center
Hampton, VA

*COMSAC Symposium
September 23, 2003*

TetrUSS is very much a team effort. This slide lists the current LaRC team members, but many have contributed in various capacities as well. We will soon launch a new and updated web site that lists many of the contributors.



The LaRC TetrUSS Team

- K. Abdol-Hamid - Turbulence modeling
- N. Frink - USM3D expert
- C. Hunter - Mac versions
- M. Pandya - USM3D expert
- P. Parikh - Large-scale applications
- S. Pirzadeh - VGRID expert
- J. Samareh - Surface geometry expert

Special thanks to many other contributors



Briefing Outline

- Overview of TetrUSS
- Typical applications
- S&C related applications
- Next steps toward S&C
- Emerging capabilities
- Summary

TetrUSS is a suite of loosely coupled computational fluid dynamics software that is packaged into a complete flow analysis system. The system components consist of tools for geometry setup, grid generation, flow solution, visualization, and various utilities tools. Development began in 1990 and it has evolved into a proven and stable system for Euler and Navier-Stokes analysis and design of unconventional configurations.

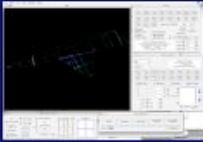
It is 1) well developed and validated, 2) has a broad base of support, and 3) is presently is a workhorse code because of the level of confidence that has been established through wide use. The entire system can now run on linux or mac architectures. In the following slides, I will highlight more of the features of the VGRID and USM3D codes.



TetrUSS

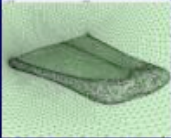
Tetrahedral Unstructured Software System

A proven, stable, and reliable multi-platform system for unstructured Euler and Navier-Stokes CFD analysis.



Geometry Setup
GridTool

- Complete flow analysis system
- Well developed and validated
- In-house experts
- Broad outside collaborations
- Workhorse system with large experience/confidence base



Grid Generation
VGRID OpenGL



Flow Solver
USM3D



Visualization
ViGLOT
(Commercial Packages)



Tools & Utilities

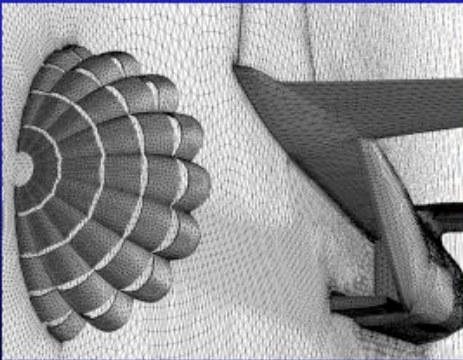
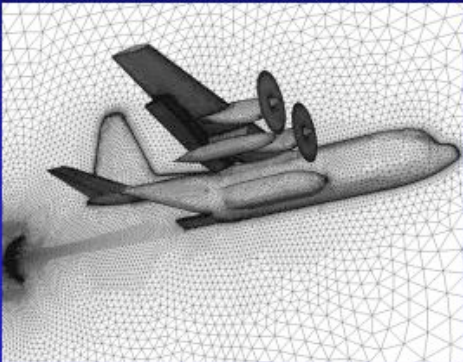
The primary features of the VGRID code are listed here. This list will mean more to the CFD savvy individuals in the audience, but the bottom line is that it will generate high-quality Navier-Stokes grids on complex geometries with a nominal amount of training. It is now quite robust and fairly easy to use.

Here is an example of a full Navier-Stokes grid generated by a U.S. Air Force Academy student (albeit a sharp student). It is a C-130 with propellers with a slotted cargo release parachute.



VGRID

Tetrahedral Grid Generator



- Thin-layer viscous tetrahedra
- Elliptically smooth grids
- Anisotropic grid stretching on Computer-Aided Design (CAD) surfaces
- Robust viscous grid movement
- Solution adaptive grid (inviscid region)
- Easy control of grid spacing
- Robust, easy to use

Sample Navier-Stokes Grid
C-130 with Cargo Release Parachute
(grid independently generated by
student at U.S. Air Force Academy)

The USM3D code is a cell-centered tetrahedral flow solver, in contrast to a “node-centered” solver for the CFD audience. It produces very accurate Navier-Stokes solutions on relatively coarse grids.

Turbulence is modeled by several models, i.e. the Spalart-Allmaras 1-eqn model, and the k-epsilon, Menter SST, and Algebraic Reynolds Stress Model (ARSM) 2-eqn models.

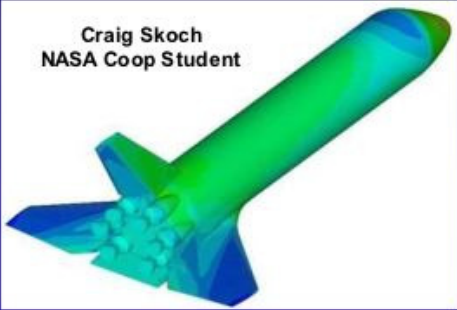
We can run both steady state with Local Time Stepping convergence acceleration, as well as unsteady flows with 2nd order time accuracy.



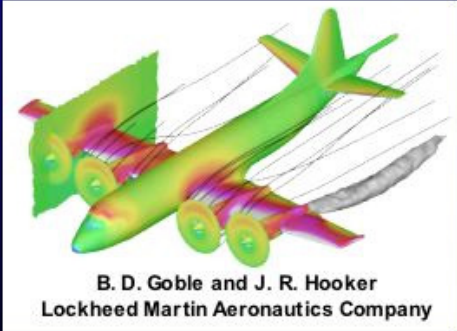
USM3D

Tetrahedral Flow Solver

- **Tetrahedral cell-centered, finite volume**
- **Euler and Navier-Stokes**
 - Several 1- and 2-equation turbulence models
- **Time Integration**
 - LTS and 2nd order time stepping
- **Upwind spatial discretization**
 - FDS, AUSM, FVS
- **Standard and special BC's**
- **Parallelized for clusters**
 - SGI, Sun, PC/Linux, Alpha/Linux, Mac OS X, IBM, HP



Craig Skoch
NASA Coop Student



B. D. Goble and J. R. Hooker
Lockheed Martin Aeronautics Company

We have a range of established upwind schemes, but have some very useful and heavily used special boundary conditions, such as propeller model, engine intakes and jet exhausts, porous surfaces, and wall functions.

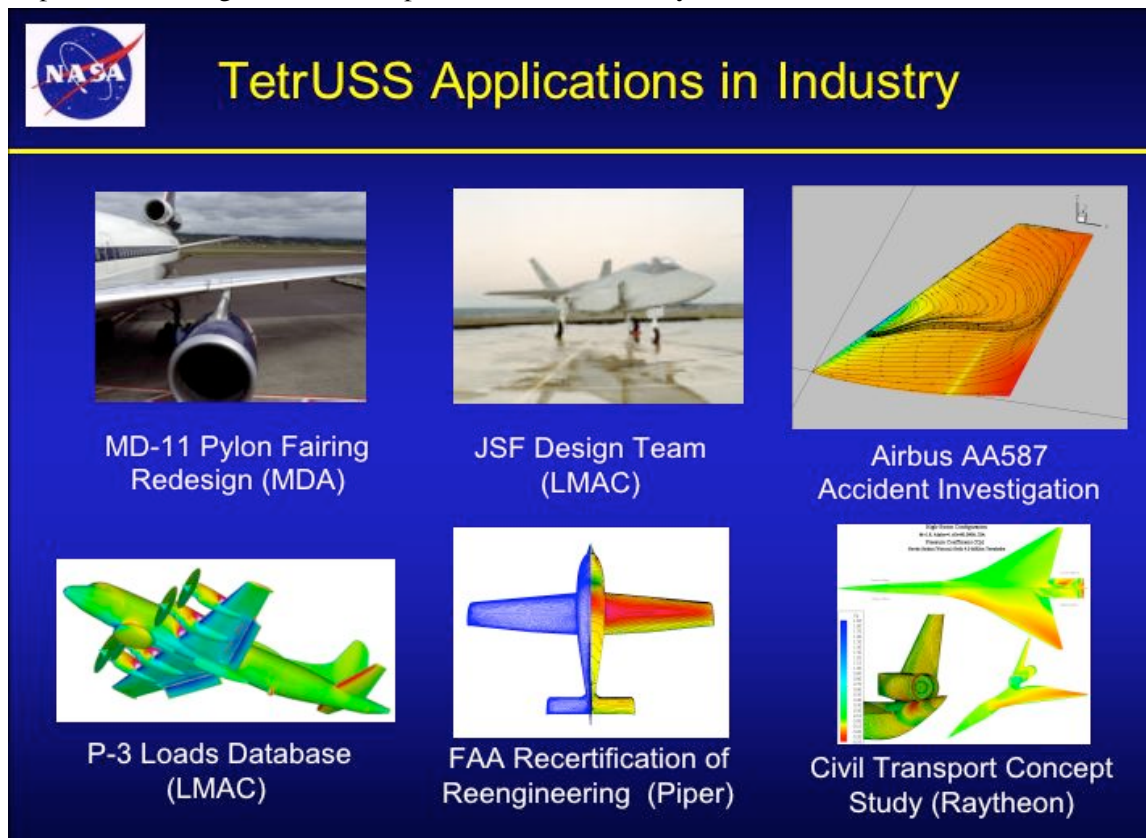
The code is very fast by current standards and runs on multiple types of parallel machine clusters.

I'll just highlight a few sample applications in industry and NASA in the next couple of slides.

Here we see a range of applications ranging from civil aircraft to fighter jets.

Back in 1992, MDA was unable to get FAA certification on the MD-11 due to a range shortfall traced to a higher than expected drag. We were requested by MDA to help eliminate an outboard pylon separation that was identified during a flight test with the expectation that it would eliminate enough drag to meet range requirements. We had a 3-month window to resolve the problem, have hardware built and flight tested. MDA sent an engineer here and we formed a tiger-team to accomplish this. The 3-month target was met and the drag was successfully reduced to permit FAA certification for range.

The JSF Design Team was another "Tiger Team" exercise with a short time scale. Several key LaRC code experts worked together to develop a new Passive Porosity BC for two structured and two



unstructured flow solvers. While I cannot show any details, the this new PassPort BC was used as an S&C tool to reduce a high-alpha pitch-up problem encountered during landing.

We were recently involved in providing some computational support to the Airbus accident investigation.

LMAC is a heavy user with many large-scale applications. Here is a sample of generating a loads database for the P-3 Orion, which required over 250 Navier-Stokes solutions on the full configuration with four co-rotating propellers.

Here is an example where Piper Aircraft has used TetrUSS to certify a reengineering of one of its aircraft.

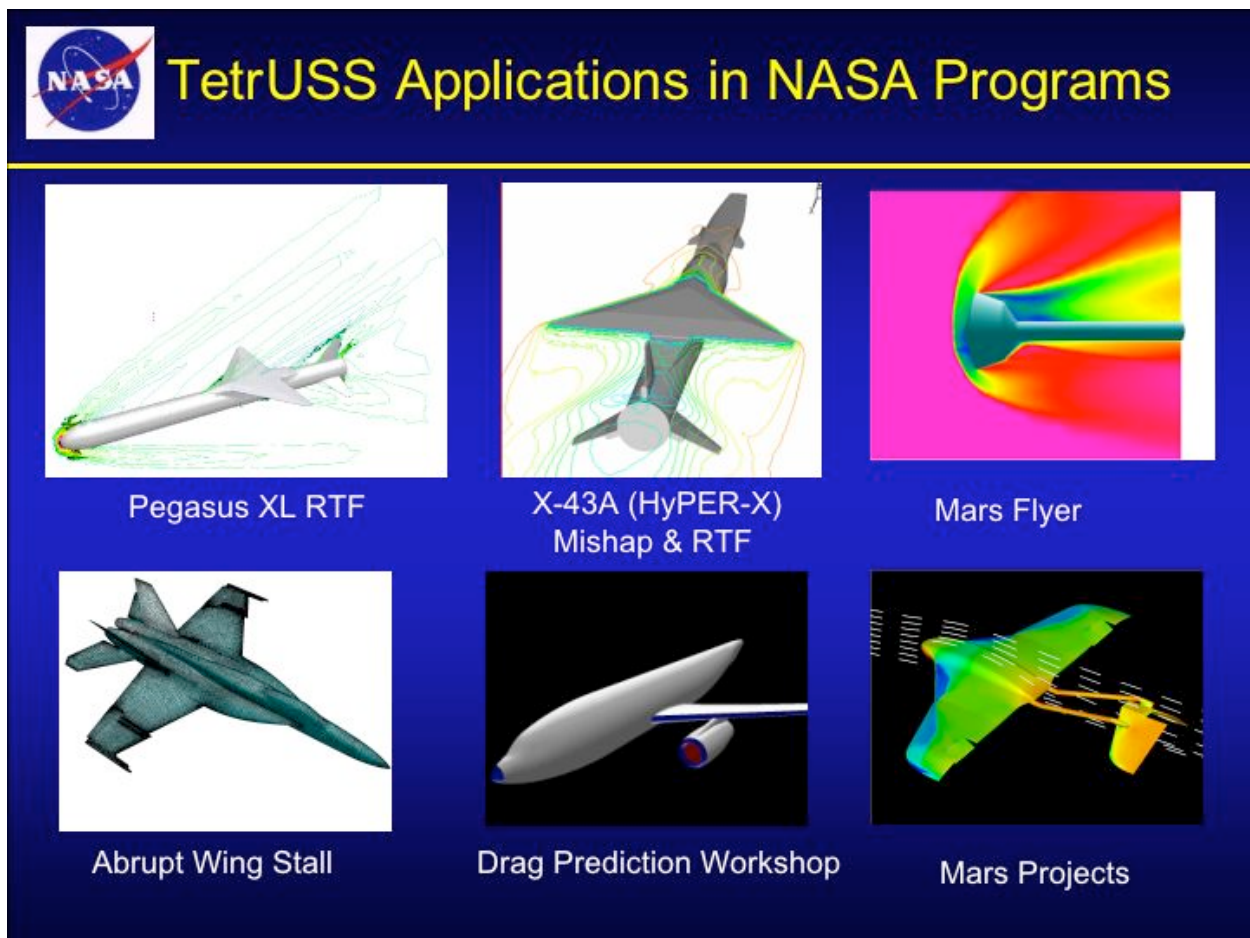
And more recently, Raytheon is using it to perform concept studies of a supersonic civil transport.

This just highlights a few of the NASA program applications.

TetrUSS was used in the mid-90's in the Pegasus Return-to-Flight effort to assess the effect of some geometry modifications on the lateral-directional stability of the new flight vehicle.

It is heavily used in the HyPER-X mishap investigation and Return-to-Flight effort. I'll show more on this in the next two slides.

TetrUSS was used in some Mars activities shown on the right.



It was heavily used in the Abrupt Wing Stall program investigating the “wing drop” phenomenon encountered on the F/A-18EF. Here massively separated flows were routinely computed and accurate results obtained.

The VGRID code was used to generate some of the unstructured grids used in the AIAA 1st and 2nd Drag Prediction Workshops, the latter held this past summer just before the Orlando Applied Aero conference. These workshops drew many participants from many countries around the world.



Overall TetrUSS Characteristics

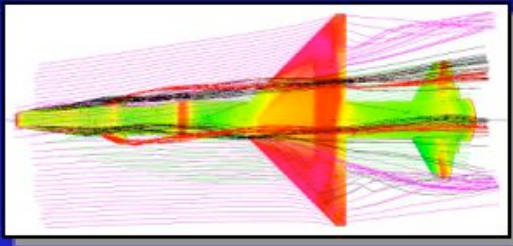
- **Ease of Grid Generation on Complex Configurations**
 - Less than one week for Euler
 - About 2 weeks for Navier-Stokes
- **Quick turn-around**
 - Navier-Stokes solution on 10-million cell grid in about 40 wall-clock hours on 48 Pentium-4 processors
- **Training class 1st week of each month includes**
 - Onsite instruction at LaRC
 - Hands-on practice
 - Free to U.S. entities

This illustrates a large-scale application of TetrUSS in response to an urgent problem. TetrUSS was the primary CFD code used in the HyPER-X Mishap Investigation and for Return-to-Flight support.

All related data is proprietary and the details cannot be discussed here, but this slide summarizes the investigation.



TetrUSS Application for S&C X-43A (HyPER-X) Mishap Investigation



- Over 60 N-S solutions on 8 separate grids over 3 month period
- Transonic flow conditions at both WT and flight Reynolds numbers
- Initial validation against WT data

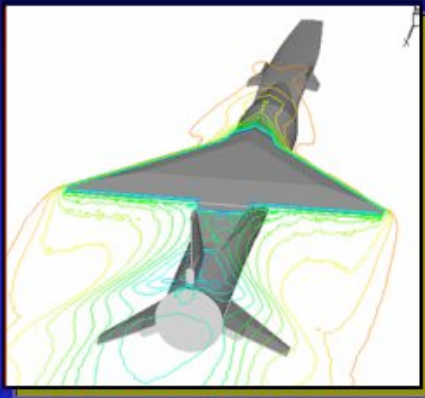
- CFD used for:
 - Filling in “gaps” in experimental data
 - Resolve discrepancies in experimental data
 - Evaluate effect of Thermal Projection System (TPS) on S&C
 - Component loads for hinge- and wing-root-bending moments
 - Aeroelastic studies

Primary Researchers: Ghaffari / Morehouse / Parlette



TetrUSS Application for S&C

X-43A (HyPER-X) Return to Flight (RTF)



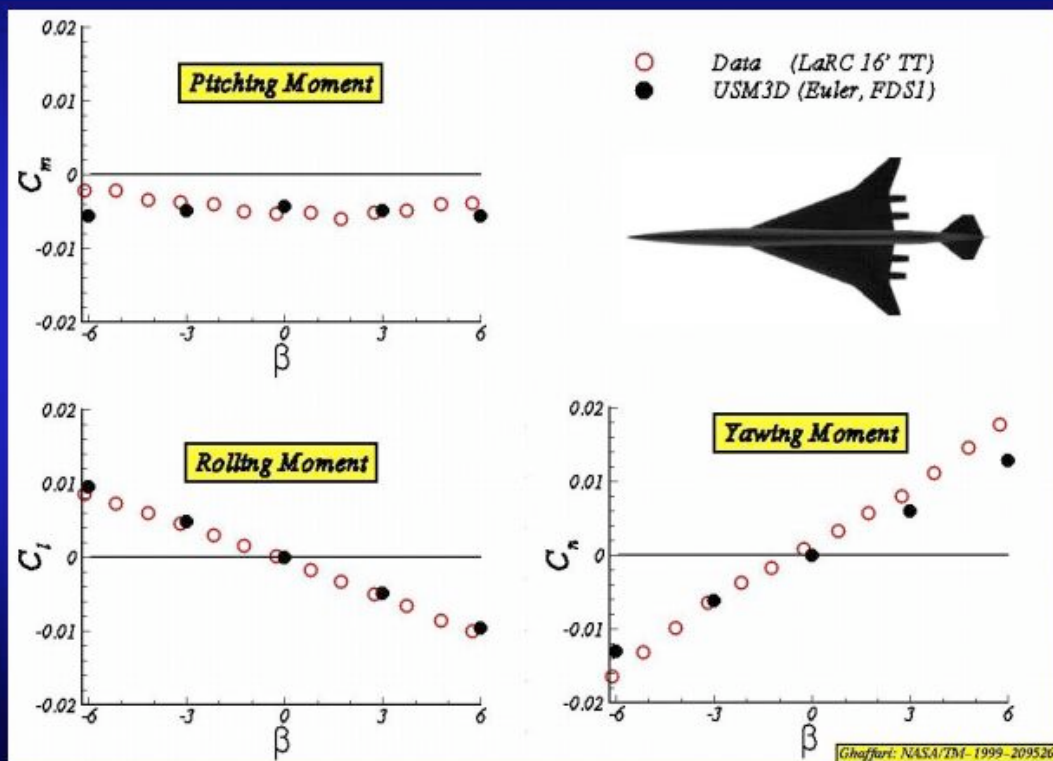
- Effort similar to the mishap investigation
- Over 60 N-S solutions on 16 different grids of full stack configuration with TPS in 3-months
- Mach range from 1.4 to 7.0
- Mostly at WT Reynolds number, some at Flight scaling

- In addition to the uses in the Mishap Investigation, **CFD was used for:**
 - Trajectory design
 - Loads and hinge moments for mechanical design of spindle & gears for the control surfaces

Primary Researchers: Armand / Tyler / Parlette / Parikh



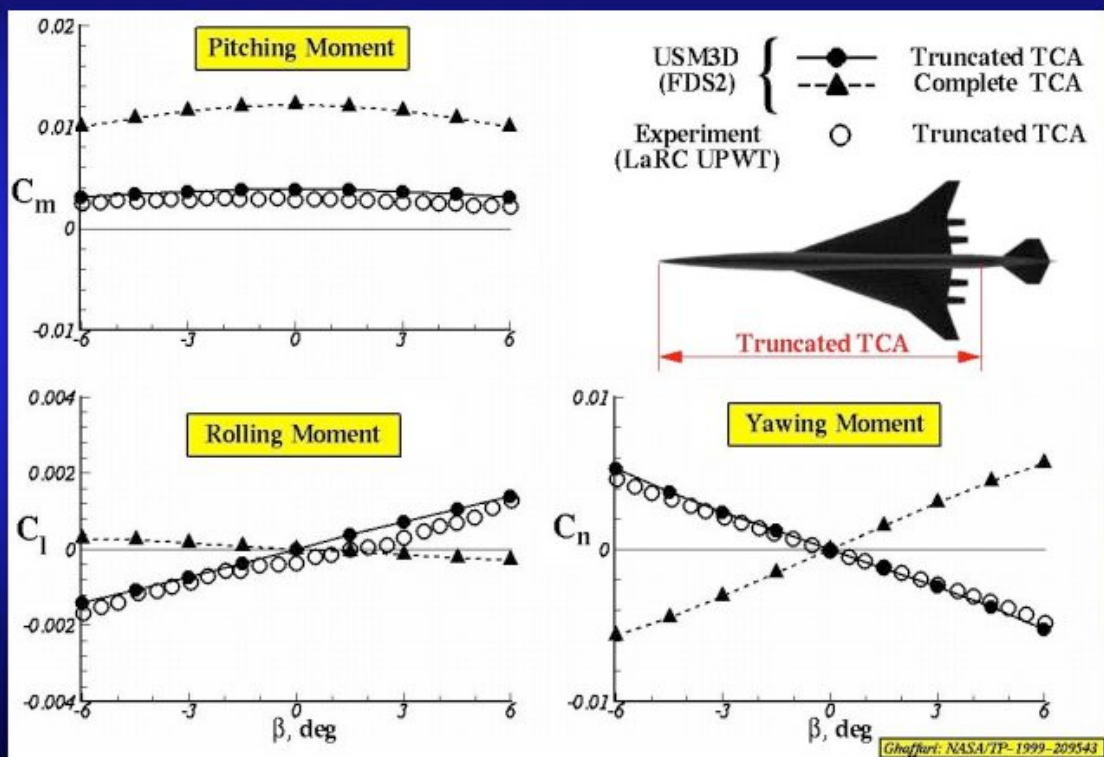
Aerodynamic Moments at Sideslip for HSR Ref-H Cruise Configurations, $M_\infty=0.95$, $\alpha=4$ deg, $Re=4 \times 10^6$





Moments at Sideslip for the TCA Configuration

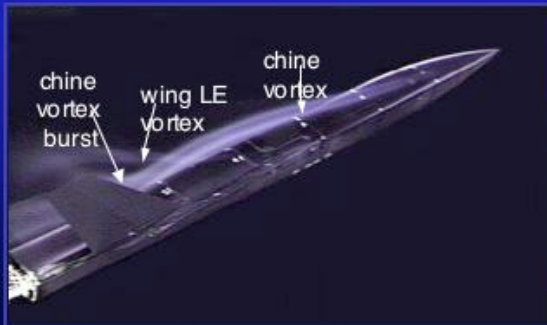
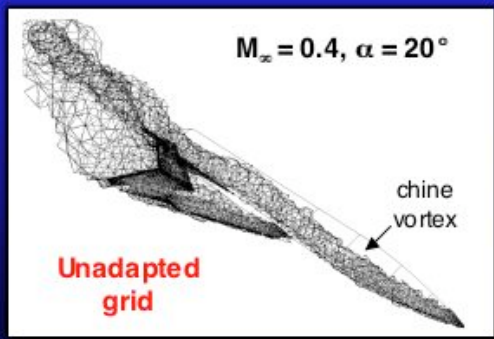
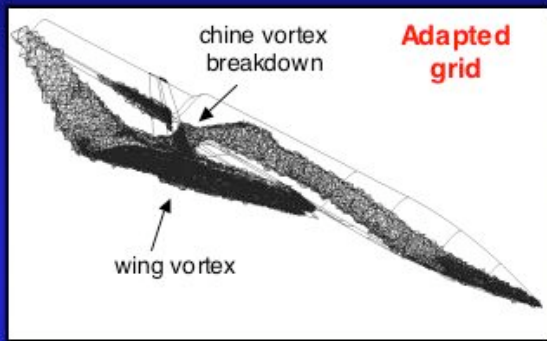
Flow-thru propulsion cruise geometry; $\alpha=0^\circ$, $M_\infty=2.4$, $Re_{ft}=4 \times 10^6$





Solution Adaptive Grid for Vortex Breakdown

- Hybrid grid adaption thru local surface refinement and volume remeshing
- Critical for correctly resolving vortex breakdown and predicting pitchup



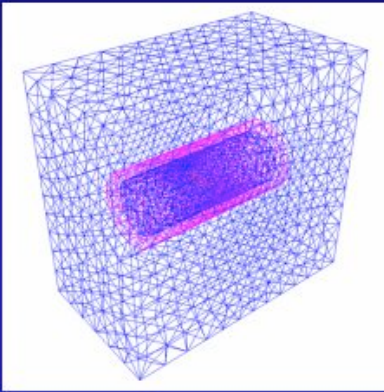


Next Steps Toward S&C

- **What we can do now**
 - Rapid grid generation
 - Large-scale computational studies
 - Solution-adaptive grid for inviscid, steady flows
 - Static longitudinal and lateral/directional derivatives
 - Massively separated flows
- **What is needed for S&C problems**
 - Dynamic longitudinal and lateral/directional derivatives
 - Dynamic grid adaption
 - Unsteady flows on moving components
 - Unsteady flow control effectors

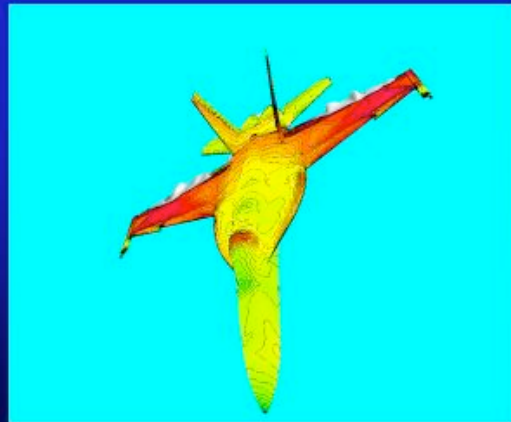


Work in Progress: Forced Oscillation



- Series of steady state solutions - one for each roll angle (φ)
- Each steady solution obtained by solving outer and inner grids alternatively, with solution interpolated in the overlapped region, until converged to SS

- An overset grid approach
- Inner grid (red) contains body and moves (oscillates)
- Outer grid (blue) is fixed
- Overlapped cylindrical regions





Future Focus toward S&C

- **Large-Scale Aero-Database Generation**
 - USM3D to be added to the **NASA Ames AeroDB** system in Fall '03
 - Enables generation of hundreds of N-S and thousands of Euler solutions per week
 - Goal is to extract simulation database and S&C derivatives
- **Chimera Overset Grid**
 - **R. Noack's DirtLib libraries** installed into USM3D in early Sept. 03 (presently under verification testing)
 - Goal to facilitate dynamic "Free-to-Roll", 6-DOF simulation capabilities and movable control surfaces
- **Detached Eddy Simulation**
 - Installed into USM3D – verification testing underway
 - Goal to include massively separated unsteady flow effects into "edge of the envelope" flight conditions



Summary

- **TetrUSS is a workhorse flow analysis system for large-scale aerodynamic problems**
 - Complete flow analysis system designed for broad application
 - Extensive validation with large user/experience base
 - Readily usable with nominal amount of training
- **Demonstrated applicability to S&C**
 - Lateral/Directional stability
 - Hinge moment estimates
- **New significant capabilities coming in near future**
 - Large-scale aero database generation
 - Chimera overset grids
 - Detached Eddy Simulation
- **We solicit your feedback on future S&C requirements**

Computational Simulations for Stability and Control ~ BCA State-of-the-Art

N. J. Yu, T. J. Kao, D. R. Bogue, and T. R. Lines
Boeing Commercial Airplanes
Seattle, Washington



COMSAC Workshop, September 23-25, 2003



Outline of Presentation

- Navier-Stokes solvers and areas of applications
- CFD issues and challenges
- Summary/future works



COMSAC Workshop, September 23-25, 2003



The presentation will show some of our recent work in using the NASA-built Navier-Stokes solvers for various applications including airplane control surface effectiveness study, Reynolds number scaling, and high lift configuration analysis.

Navier-Stokes Solvers and Areas of applications

- CFL3D/TLNS3D (NASA Langley)
 - ~ Spoiler reversal study
 - ~ Outboard aileron effectiveness
 - ~ Stabilizer/elevator effectiveness
 - ~ High speed pitch characteristics with vortex generators
- OVERFLOW (NASA Ames/Langley)
 - ~ High lift configuration
 - ~ Slotted wing

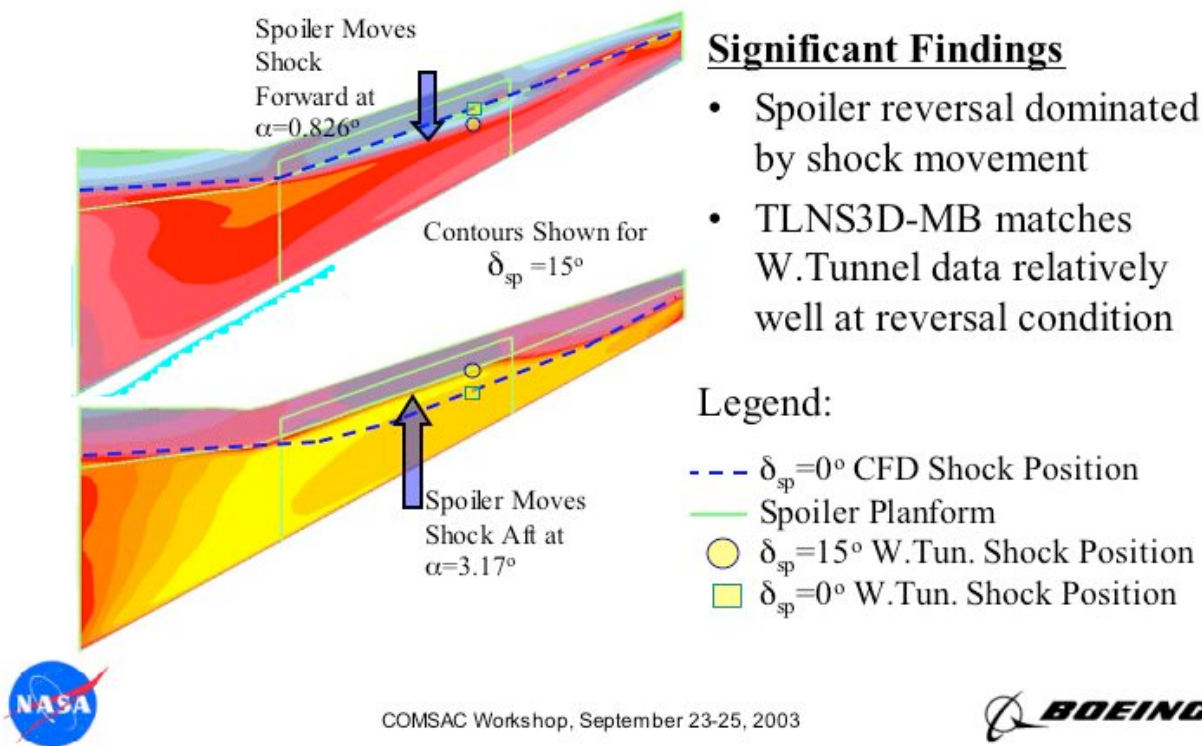


COMSAC Workshop, September 23-25, 2003



Our first application of Navier-Stokes technology to S&C problem is the use of TLNS3D code to predict the high speed spoiler reversal phenomenon. At high transonic Mach with given spoiler deflection, the wing lift decreases at low angle of attack, as expected. However, as alpha increases beyond a certain value, lift increases; a phenomenon known as spoiler reversal observed in the wind tunnel test. CFD not only gives correct prediction, it also provides flow field details which explain the cause of spoiler reversal.

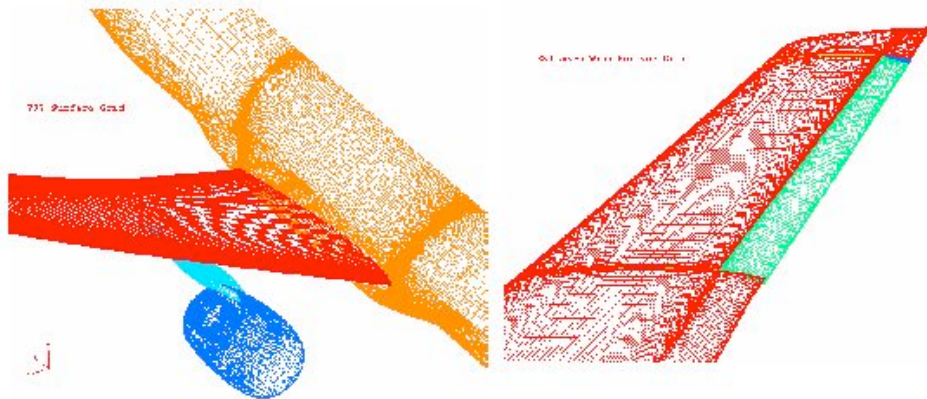
High Mach Spoiler Reversal



More recently, we have used both TLNS3D and CFL3D code to evaluate outboard aileron effectiveness with respect to Reynolds numbers. Multiblock grid approach was used to provide detailed representation of the geometry including aileron gaps. Both flow through and powered nacelles can be simulated in the analysis. Preliminary results show that effects of Reynolds number on aileron effectiveness were predicted correctly. More detailed study is in progress.

Twin Engine Configuration with Deflected Outboard Aileron

Wing/Body & Strut/Nacelle Model

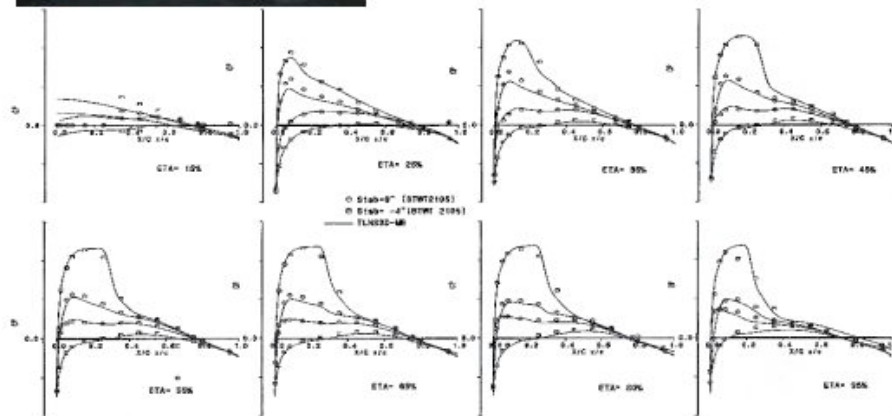
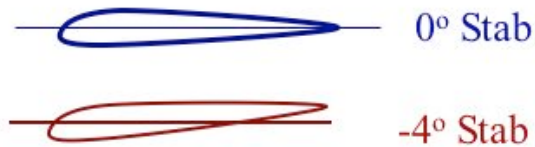


COMSAC Workshop, September 23-25, 2003



Applications of CFD for stabilizer or elevator effectiveness prediction are more challenging than the aileron analysis. One needs to resolve and capture the wakes generated by the wing, fuselage and nacelle, as they can affect the flow field in the horizontal tail region. With the use of denser grid in the wake regions and in the horizontal tail region, the CFD results correlate well with wind tunnel tail pressure data, and the tail lift slope curve also correlates with NTF data reasonably well.

High Speed Stabilizer Effectiveness

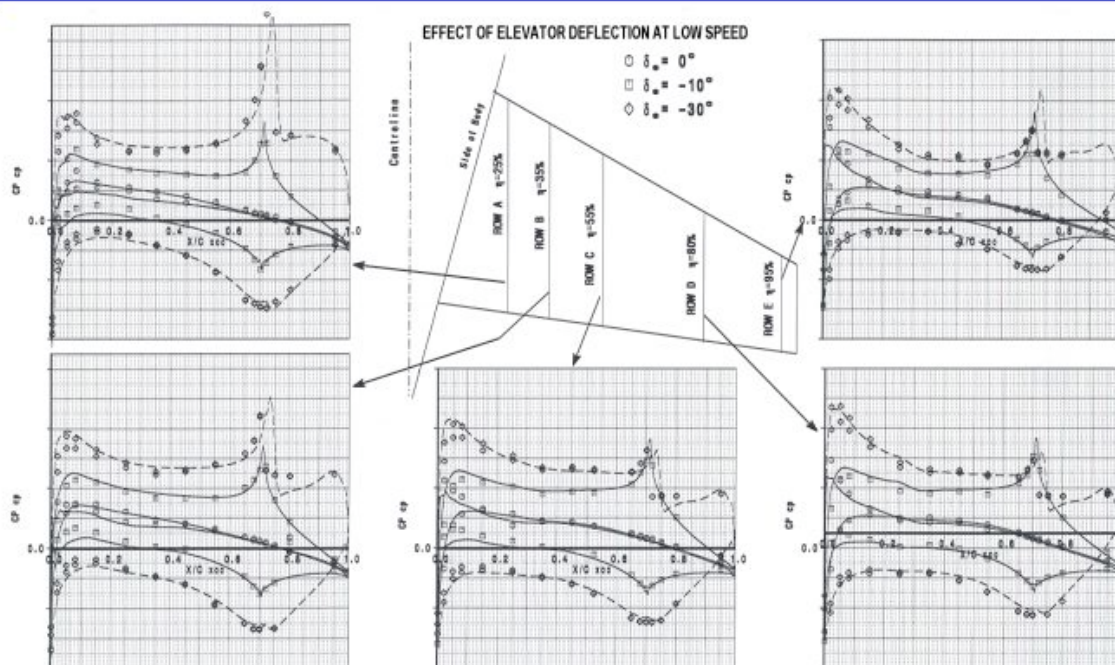


COMSAC Workshop, September 23-25, 2003



Similar analyses were carried out for elevator effectiveness prediction. Here the elevator deflection up to -30 deg. was analyzed. Even with fairly massive flow separation downstream of the elevator hinge line, the CFD analysis with Menter's two equation SST turbulence model provides good correlation with test data.

Test/theory comparisons for elevator analysis



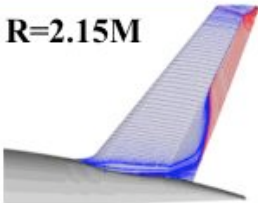
COMSAC Workshop, September 23-25, 2003



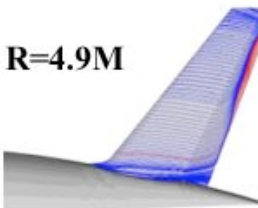
More extensive analyses were carried for low speed elevator Reynolds number effects. Boundary layer on the elevator becomes healthier at higher Reynolds number for a given elevator deflection. The flow separation downstream of the elevator hinge line at low Reynolds number reduces significantly at higher Reynolds number, and thus increases elevator effectiveness. The CFD prediction on elevator Reynolds number effects correlates reasonably well with the data.

Low Speed Elevator Reynolds Number Effects

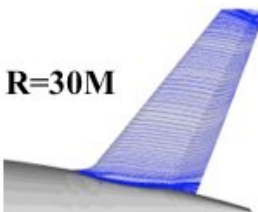
R=2.15M



R=4.9M

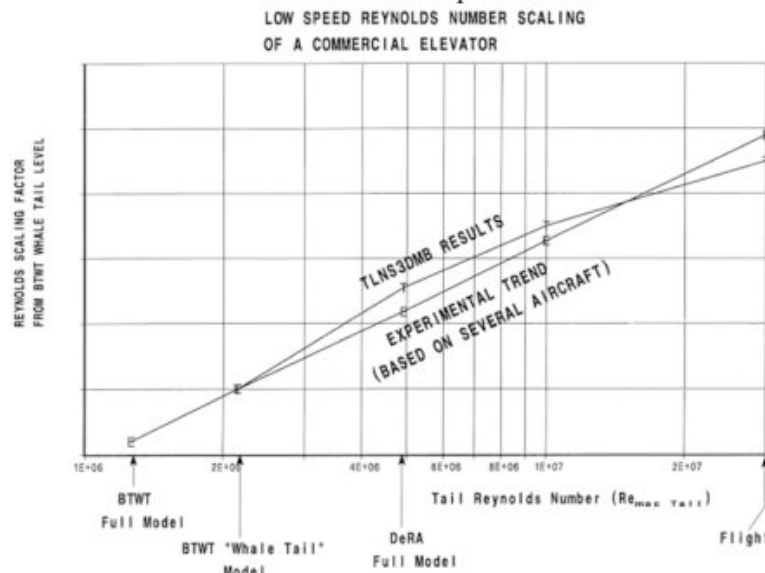


R=30M



Findings:

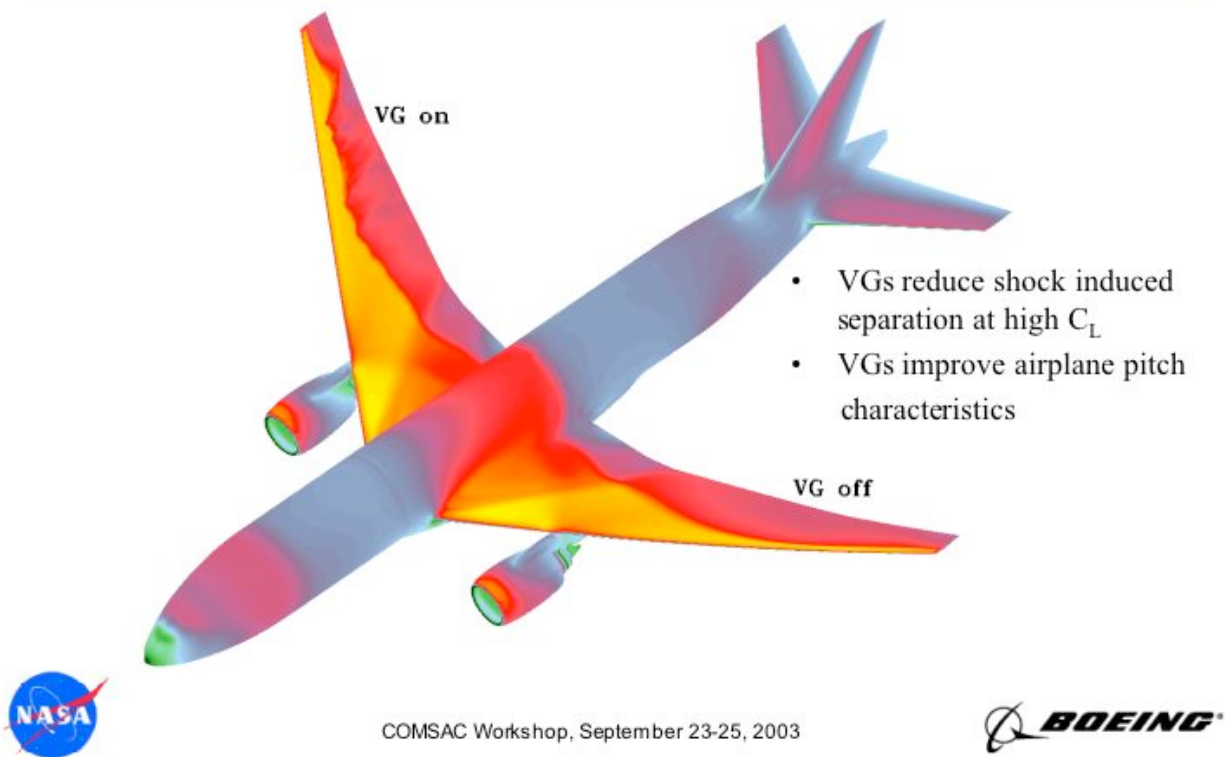
- TLNS results correlate well with Experimental trends



COMSAC Workshop, September 23-25, 2003

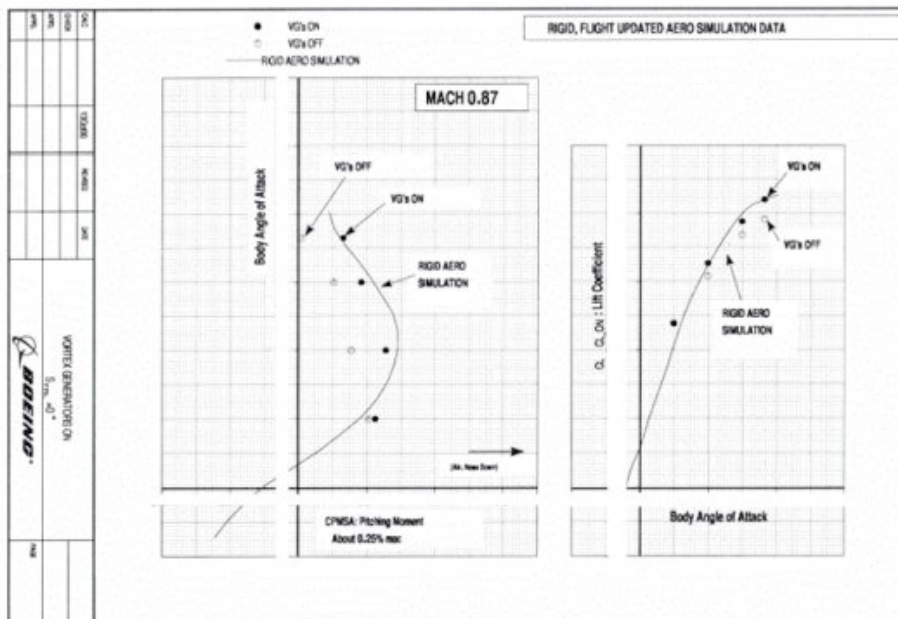


Effects of Vortex Generators (VG) on Wing Pressures at High Transonic Mach & High Alpha



For structured, multiblock grid approach, the simulation of a cruise airplane configuration including all vortex generators on the upper surface of the wing is a significant challenge to both the grid generation and flow analysis tasks. To capture flow field details generated by the vortex generators, one must use fine grid in the vortex generator regions, and to capture and preserve the vortical flows downstream of the vortex generators, accurate Navier-Stokes solver with minimum numerical dissipation is needed. A patched grid system was used in the present study. Results showed that the effects of vortex generators on shock movement, and on airplane pitch characteristics were correctly predicted. The computing resources for this analysis are fairly reasonable, using 56 CPUs of an SGI Origin machine, one can get the solution of one flow condition (with 25 million grid points) within about 11 hour flow time.

VG Effects on Force & Moment Data at High Transonic Mach



ANALYSIS CASE

- Wing/ Body/ Nacelle/ Strut/ Tails + 16 VG's
- No Nacelle Chine
- Analyzed at flight condition (Cruise power)
- Flight Re

FINDINGS

- VG's effective at high alpha
- VG on results correlate well with flight test data



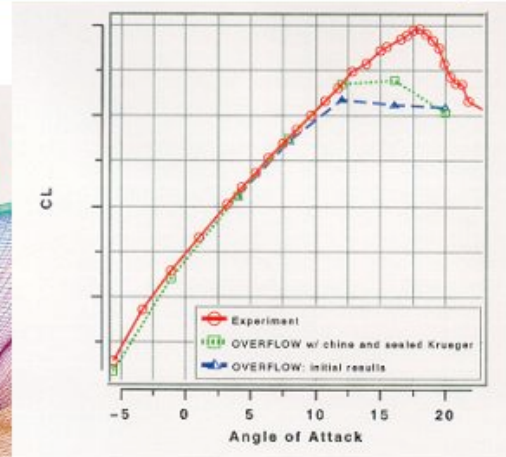
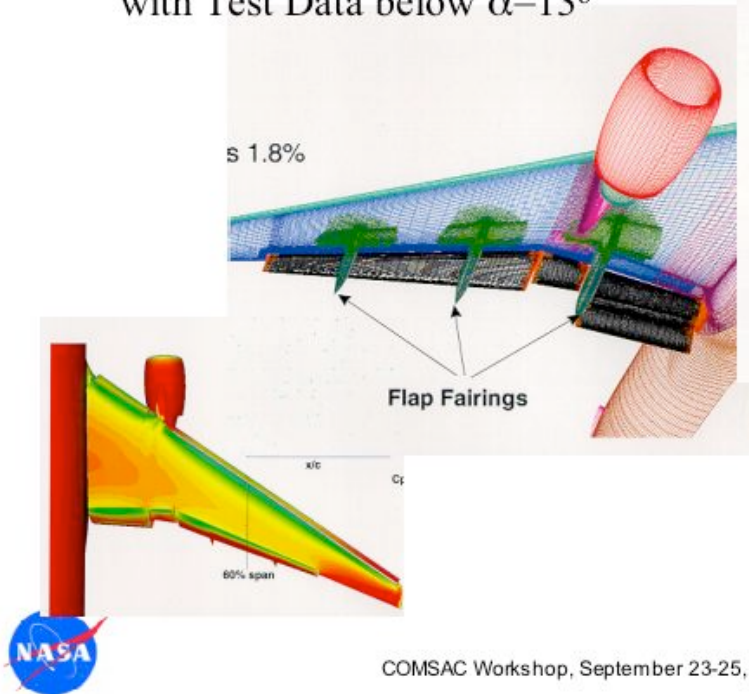
COMSAC Workshop, September 23-25, 2003



The most extensive use of OVERFLOW within Boeing Commercial Airplanes is the high lift analysis. Due to geometry complexity, overset grid is a preferred method for high lift predictions. At angle of attack lower than 13 deg., OVERFLOW results agreed quite well with test data. Above 13 deg., significant flow separation occurs on the flap elements, CFD results showed premature drop in lift. The causes of such discrepancy could be grid resolution, or turbulence model effects, which will be addressed in the later slide. More recently, OVERFLOW was also used heavily for high speed configuration analysis and design.

High Lift Analysis

- Overflow Results agree well with Test Data below $\alpha=13^\circ$



- Premature Stall above 13deg (Separation Dominated)

COMSAC Workshop, September 23-25, 2003



CFD Issues/Challenges

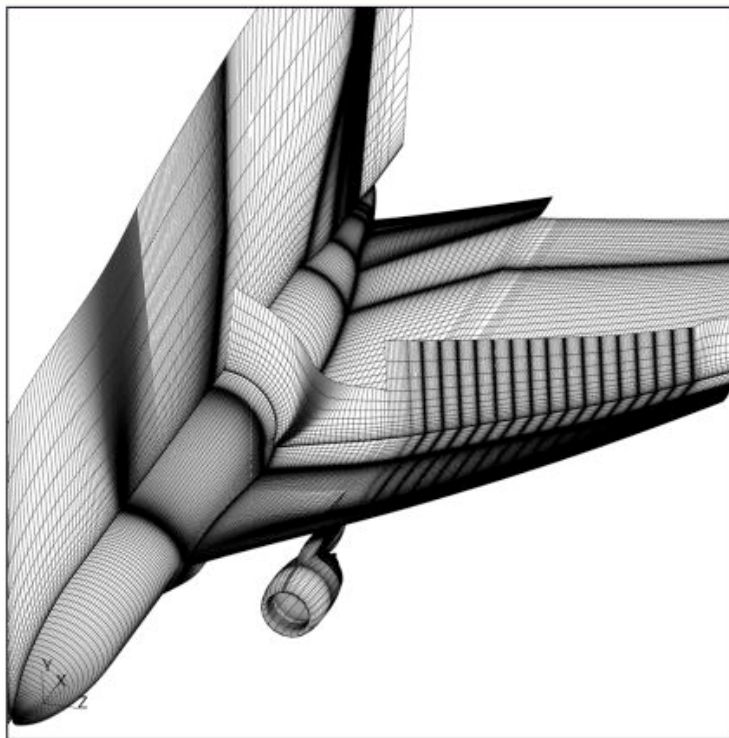
- **Geometry/surface grid & flow field grid generation**
- **Turbulence modeling for separated flow**
- **Transition location, trip effects, wind tunnel wall (slotted) simulation**
- **Algorithm issues ~ accuracy, robustness, efficiency, etc.**
- **Large Caseload Capability**
- **Unsteady Flow, including dynamic derivatives**



COMSAC Workshop, September 23-25, 2003

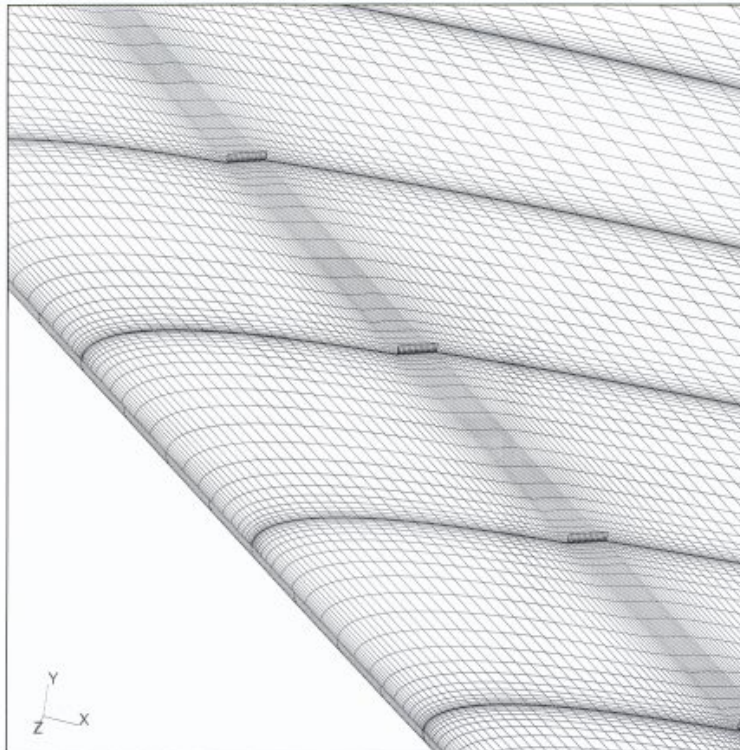


Geometry & Grid Generation – Twin-Engine Airplane with Vortex Generators



For complex geometry analysis, grid generation is one of the most time consuming part of the analysis process. Grid spacing and grid quality affect numerical solution, especially for Navier-Stokes analysis. We have spent significant efforts in the past few years to develop both surface grid and flow field grid generation capability for either structured multiblock grid or overset grid. For the Navier-Stokes codes we are using at the present time, good quality grid is essential to get accurate and reliably converged solutions. The slides show the patched grid system used in the vortex generator simulation.

Geometry & Grid Generation – Detailed Surface Grids near Vortex Generators



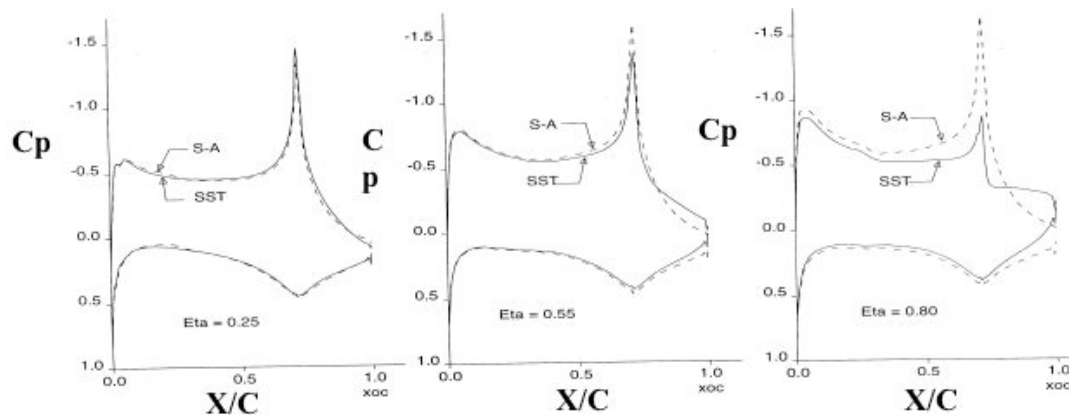
For cruise configuration analysis at near design conditions, most turbulence models can provide results which correlate well with test data. However, problems related to S&C or Loads applications usually have significant separations in the flow fields, where different turbulence models could give rather different results. The plots showed that at large elevator deflection (-30 deg.) where massive separation occurred downstream of elevator hinge line, the two-equation SST model provides more realistic results than the one-equation S-A model.

Turbulence Model Study

Menter's k- ω SST vs Published SA

Findings:

- Minor differences for deflections of 0 and -10°
- Menter's model results in increased outboard separation at the -20°
 - Separation correlates with data better than SA model results
- Menter's model converges better at -30° deflection
- Menter's model used for the remainder of this study



COMSAC Workshop, September 23-25, 2003



Transition, Slotted Wall Simulations, etc

- Boundary layer transition simulation ~ Essential for accurate viscous drag prediction, and for high lift analysis
- Slotted wall simulation ~ may be crucial for unsteady loads validation



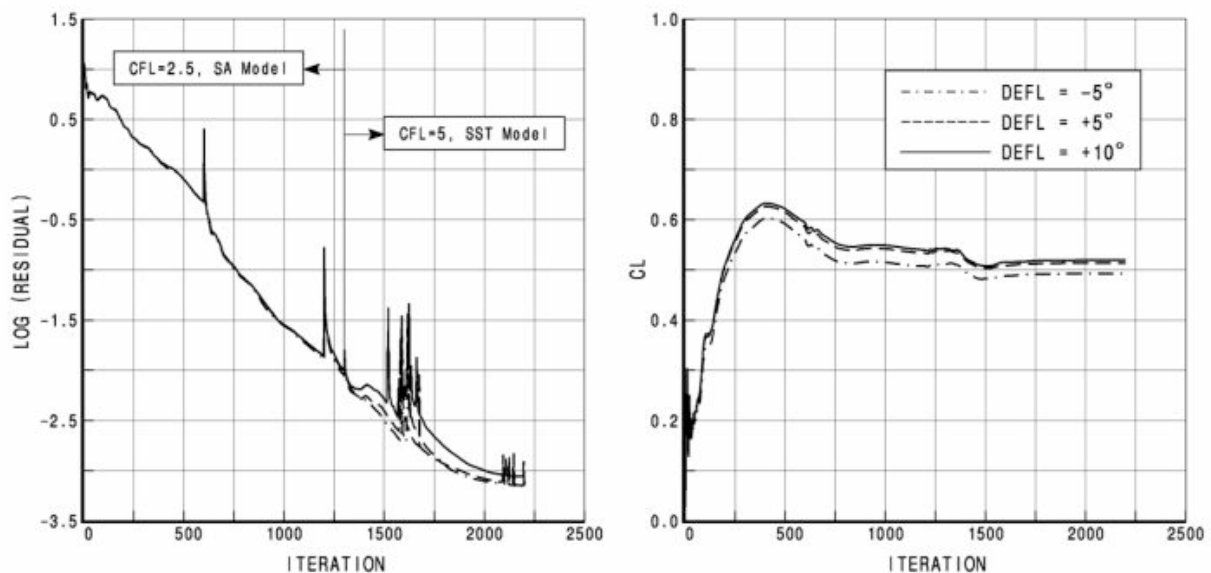
COMSAC Workshop, September 23-25, 2003



Reliable convergence for N-S analysis is essential for S&C or Loads applications, where large number of cases are needed within a short period of time. The plots show typical convergence of CFL3D code at transonic speed. It requires many hundreds of multigrid cycles to converge the solution to acceptable level of numerical accuracy. For some cases, the convergence history could be worse, which deserve further research and improvement.

Algorithm Issues

Convergence History for a W/B/N/S with Deflected Aileron



COMSAC Workshop, September 23-25, 2003



Large Caseload Capability

- Reynolds number scaling study
~ Problem size: 25 million grid points
- Wing control surface effectiveness study
~ Problem size: 30 to 65 million grid points
- Empennage analysis
~ Problem size: 10 to 15 million grid points
- High Lift Analysis
~ Problem size: 30-100 million cells

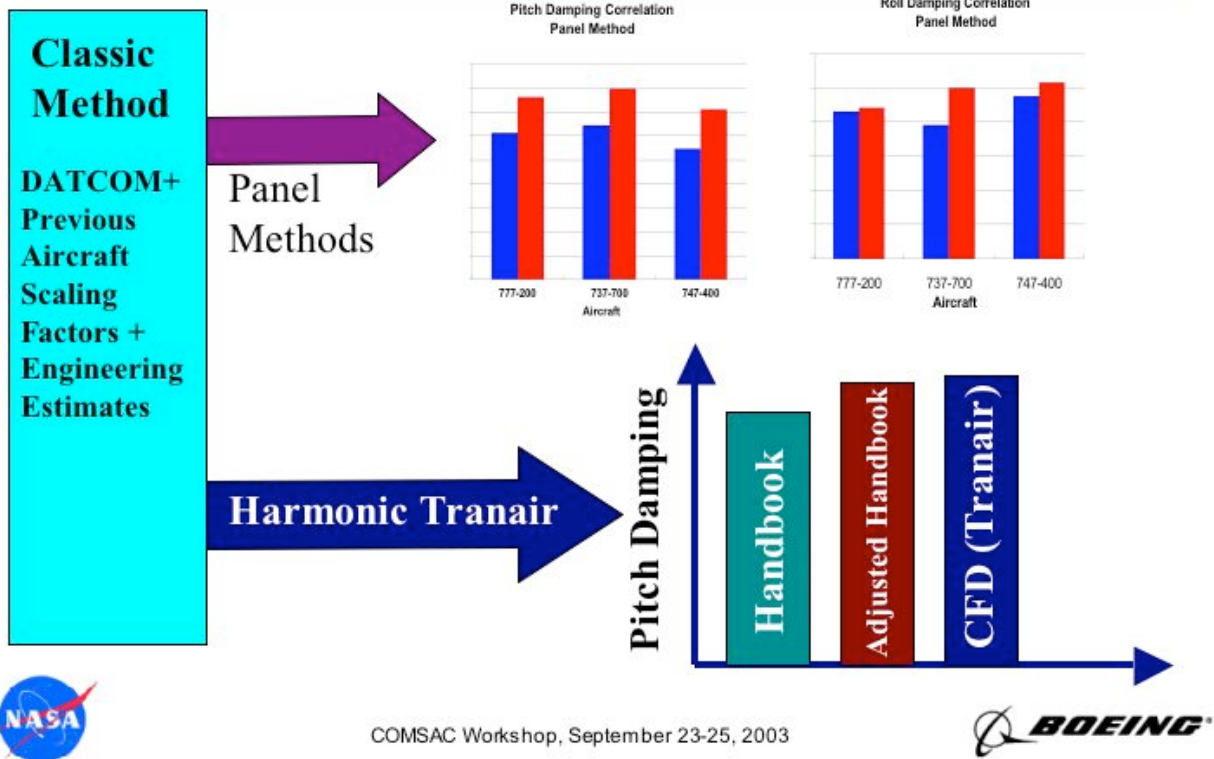
Each category needs hundreds of runs



COMSAC Workshop, September 23-25, 2003



Dynamic Derivative Prediction Methods



Summary/Future Works

- Promising CFD results for S&C applications
- Geometry/grid generation for complex configuration is still the most time consuming part of the process ~ major improvements & automation needed
- Turbulence models inadequate for separated flow simulation ~ require continuous improvements
- Solvers robustness/efficiency need improvements for complex flow simulations encountered in S&C applications



COMSAC Workshop, September 23-25, 2003



Acknowledgements

- Boeing Commercial Airplanes
 - ~ CFD application group
 - ~ Product development : S&C, Loads groups
 - ~ Product support group
- NASA Langley/Ames Research Centers
 - ~ TLNS3D, CFL3D support
 - ~ OVERFLOW support



COMSAC Workshop, September 23-25, 2003



Abstract:

Time accurate CFD may offer a faster approach to S&C aerodynamic database population than the conventional point by point steady state CFD. We would directly simulate α -, β -sweeps or other configuration movements typically of measurement sequence in wind tunnels. A second objective is to demonstrate potential applications to assessment of S&C dynamic derivatives by simulating vehicle motions such as free to roll, and nonlinearity such as the trends of aerodynamic forces near CL-max or flow hysteresis.



Time-Accurate Computational Aerodynamic Simulation

S. Paul Pao and Pieter G. Buning

Configuration Aerodynamics Branch
NASA Langley Research Center

COMSAC Symposium
Hampton-Virginia, September 24, 2003



Outline

- Unsteady Motion simulation:
 - Oscillating pitch motion
 - β - sweep at finite rate
 - Periodic body axis roll motion at $\nu = 0.05$
- Database population by continuous sweep
- Summary and future work



Why Time-Accurate CFD

- Simulate unsteady motion, maneuver, and aero-elasticity
- Provide an alternate approach for efficiency static S&C database population
- Can provide high density continuous data which may not be practical to obtain by steady-state CFD
- Can separate “lumped” dynamic derivatives by direct CFD motion simulation of the relevant components
- Offer a better quantitative assessment of S&C coefficients when the situation involves flow separation

Moving grid algorithm must be efficient and robust for the user. The specifics include simple I/O, maintenance of good grid quality throughout thousands of computational cycles, and fast grid transformation such that grid motion time is a fraction of the time required for a single Navier-Stokes iterative step.



CFD Methodology (Structured Mesh)

- Fast and robust deformable moving grid algorithm written for structured mesh computations
- Present capability is limited to prescribed motion simulation
- Used the OVERFLOW code and SA turbulence model in this study
- 3D configurations with 2 million grid points took approximately 160 CPU hours per case

For obvious reasons, the helicopter aerodynamics research community had produced some of the most comprehensive time-accurate wind tunnel measurements and CFD analysis. We have chosen the original data by Piziali and the associated CFD studies as our reference for comparison.



Time-Dependent Wind Tunnel Data & CFD Simulations of WT Cases

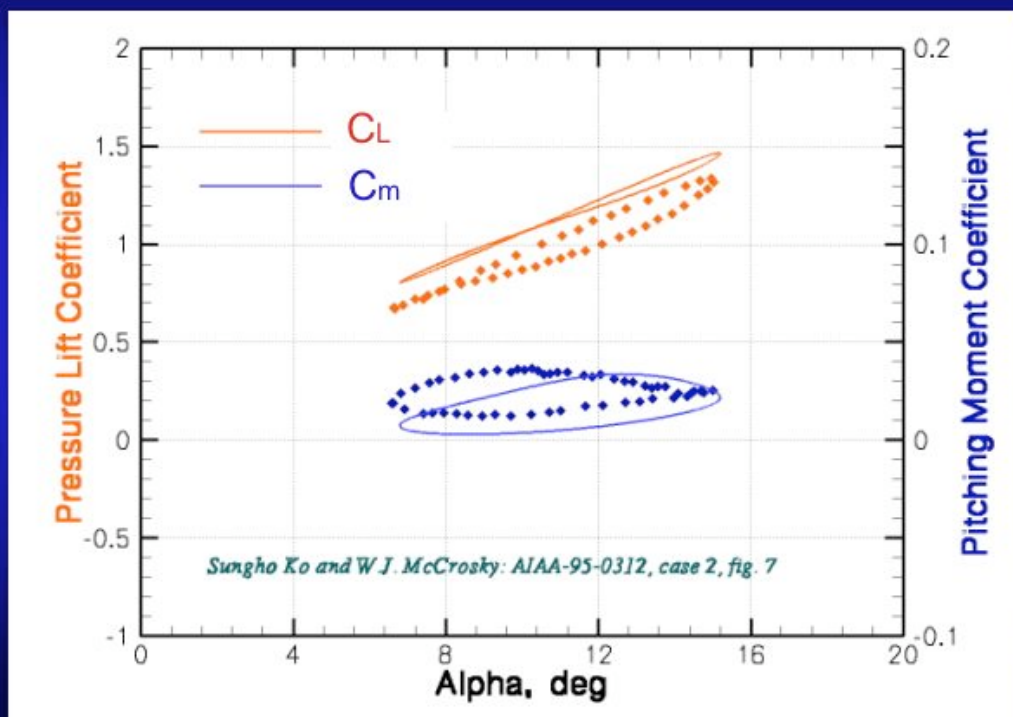
- US Army/NASA Ames oscillating wing data (cyclic helicopter blade WT simulation) by Piziali, 1994
- CFD simulation for selected 2D cases by:
 - Ko and McCroskey: 1995
 - Sankar, Zibi-Bailly et al. : 2002
- NACA-0015 Airfoil: 12 inch chord, “infinite” span (2D) and aspect ratio =5 semi-span WT model.
- Mach number = 0.290; Reynolds number = $1.95\text{E}+6$

Although the lift and pitching moment comparison are close, there are distinct differences between CFD and measurement. For example, the lift coefficient is higher than the measured data and the hysteresis loop is narrower for the CFD solution. The loop shape of the pitching moment are also different. However, this is typical for comparisons between CFD results using several other codes and the experimental data by Piziali.



Oscillating Airfoil: Lift and Pitching Moment

Data and CFD : $\alpha = 11 \pm 4.2$ deg

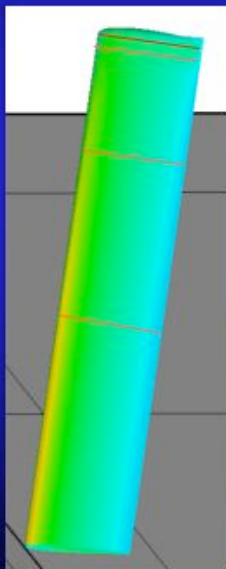


Three dimensional unsteady CFD simulation is rarely available in the literature on account of the computational expenses and the difficulties in obtaining converged solutions. These are not fundamental obstacles and we should see much more often applications of 3D time accurate to practical problems in aerodynamics and S&C in the near future. This example demonstrates the distinct change in aerodynamic behavior in the spanwise direction. The pitching moment sign and slope near the wing tip are different from those for all the inboard stations.

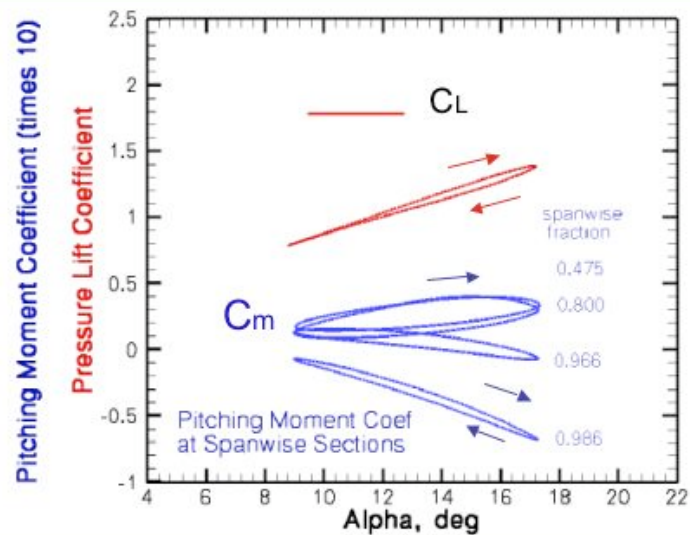


NACA-0015 Semi-Span Wing Pitching at 13 +/- 4.2 Deg

Pitching Moment Loci at Different Spanwise Sections



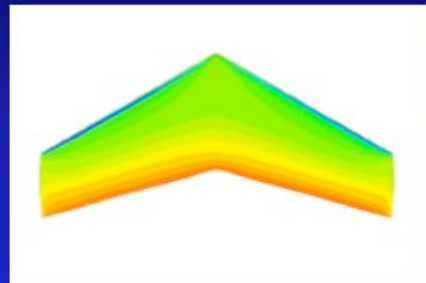
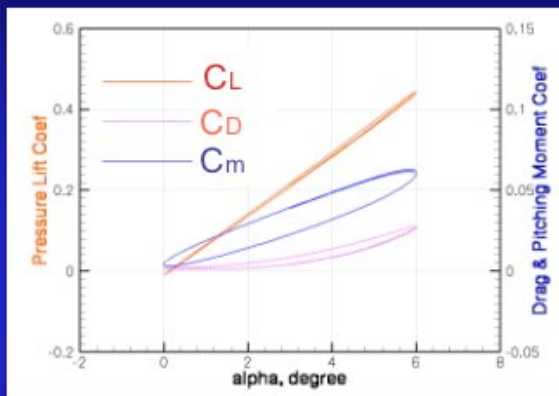
Y/B = .986
Y/B = .966
Y/B = .800
Y/B = .475



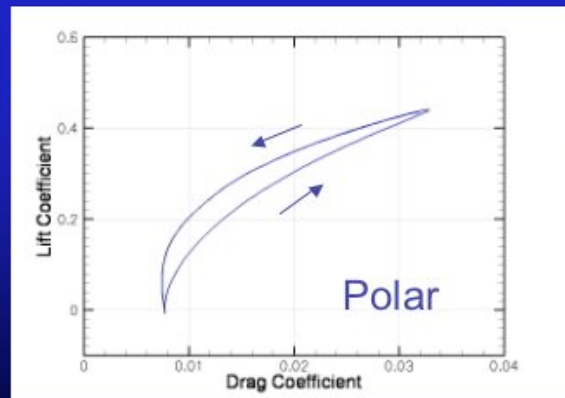
Although we don't have data to compare with, this is to demonstrate feasibility a continuous α sweep at a high subsonic Mach number. The lift and drag coefficient information is also plotted as a drag polar. The nondimensional time of $T=210$ for the angle of attack to go from 0 to 6 deg and back is relatively fast: less than one second in terms of wind tunnel length and time scales.



ONERA-M6 Full Span Wing α - Sweep



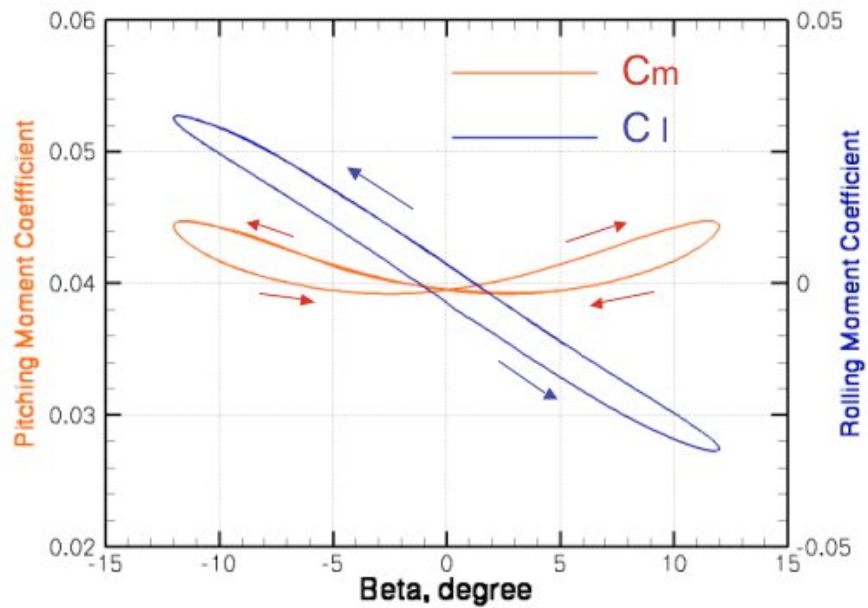
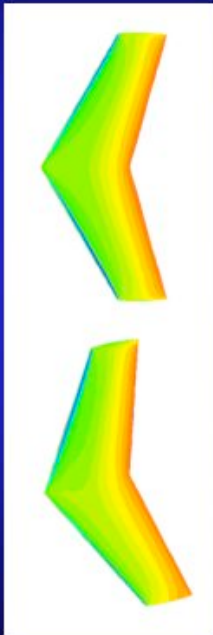
- Originally a semi-span model
- Mach number = 0.70
- Reynolds number = $11.7E+6$
- Cycle time = 210



The subject of interest for this finite rate beta sweep simulation is the rolling moment coefficient. The figure shows a narrow hysteresis loop with nonlinear slopes of C_l -beta for beta greater than about ± 8 degrees.



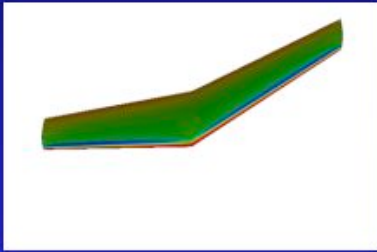
Beta- Sweep: $\beta = \pm 12$ deg; $\alpha = 4$ deg



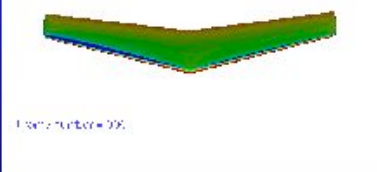
The rolling moment response of the ONERA-M6 wing to a periodic body axis roll motion is classical. From this C_l versus Φ locus, we can determine the roll damping dynamic derivative for the given roll rate from the slopes of the curve at $\Phi = 0$.



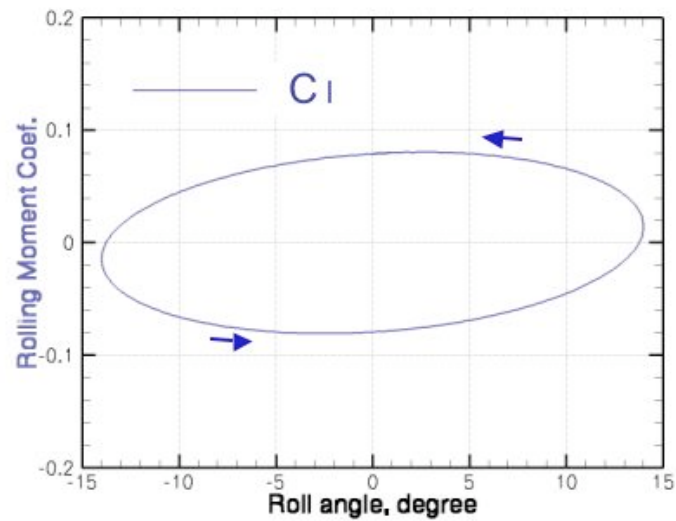
Body Axis Roll : $\phi = \pm 14$ deg ; $\alpha = 3$ deg



ONERA M6 wing geometry
Roll axis: $\phi = \pm 14$ deg ; $\alpha = 3$ deg



Roll rate: 0.05



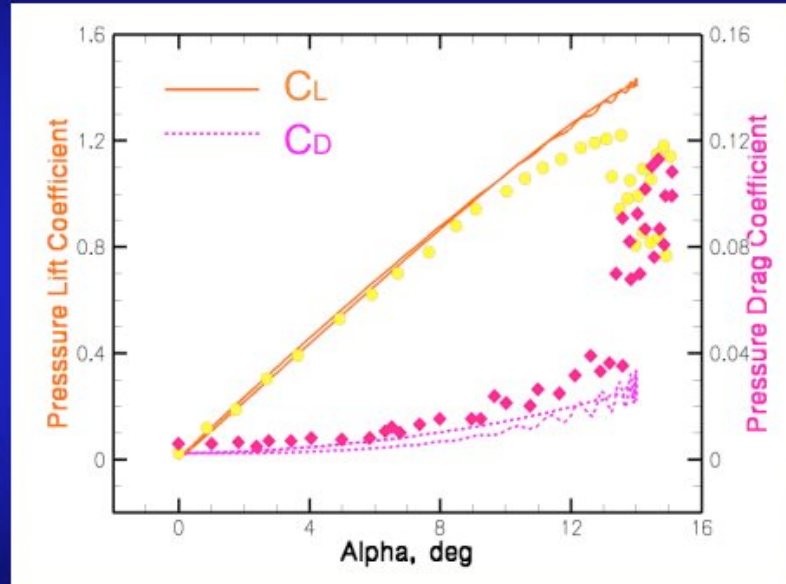
The nondimensional cycle time needs to be sufficiently large for steady-state simulations such that hysteresis due to motion dynamics is not present. A sequence of calculations at different cycle time ranging from 100 to 4000 indicated that cycle time above 1000 will do well. On the other hand, the time step size is governed by flow physics. The nondimensional time unit is the time required for a fluid particle to travel the distance of one chord. At $DT=0.2$, we are tracking a fluid particle only five times over the airfoil as it moves nominally according to the prescribed free stream velocity. As a result, a complete cycle required 20000 time steps. From a CFD point of view, it is equivalent to computing 10 independent steady state solutions for the same wing configuration.



2D Airfoil α - Sweeps: 0 - 14 Deg

Asymptotic approach to steady-state

- Steady state approximated by very large cycle time
- Dt limited by flow physics & CFD convergence



$T_{\text{cyc}} = 4000$, $Dt = 0.20$; (Physical cycle time = 10 sec)



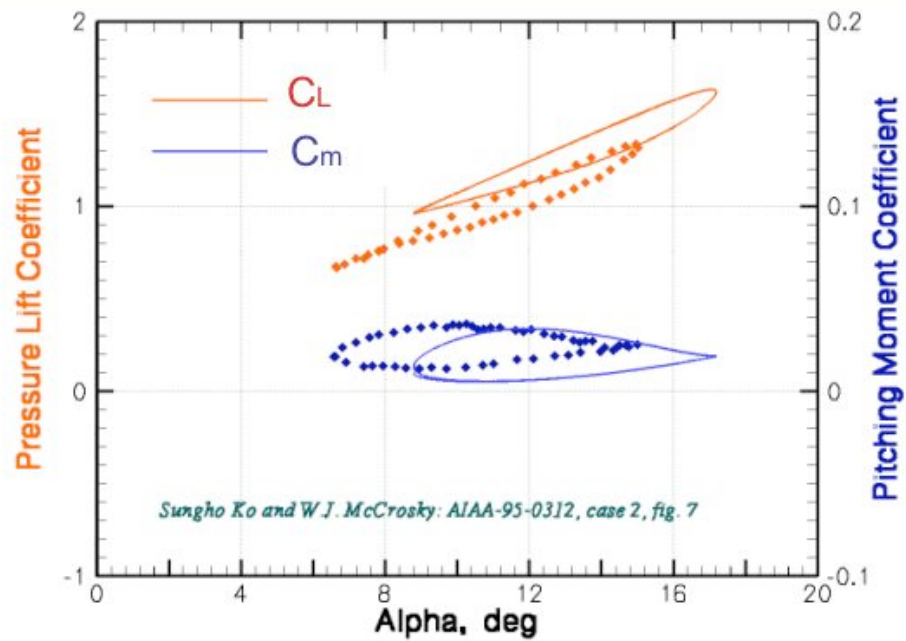
Interesting CFD Observations

- α - shift of forces and moments between CFD and WT measurement
 - numerical algorithm and turbulence model
 - WT measurement assumptions
- α - sweep through NACA-0015 stall region
 - mimic measurement uncertainty in C_L , C_D
 - turbulence model may affect $C_{L\max}$

The computed lift and moment coefficient curve shapes are now practically the same. One of the explanations could be that the CFD flow separation is delayed by 2 degrees in α versus the wind tunnel experiment. At this time, it is not entirely clear what is the cause of this discrepancy. The experiment was done with an infinite span wing, not a true two dimensional representation. Further investigation could repeat this simulation using an infinite span CFD configuration. Instead of URANS, we may have to use DES or other hybrid techniques.



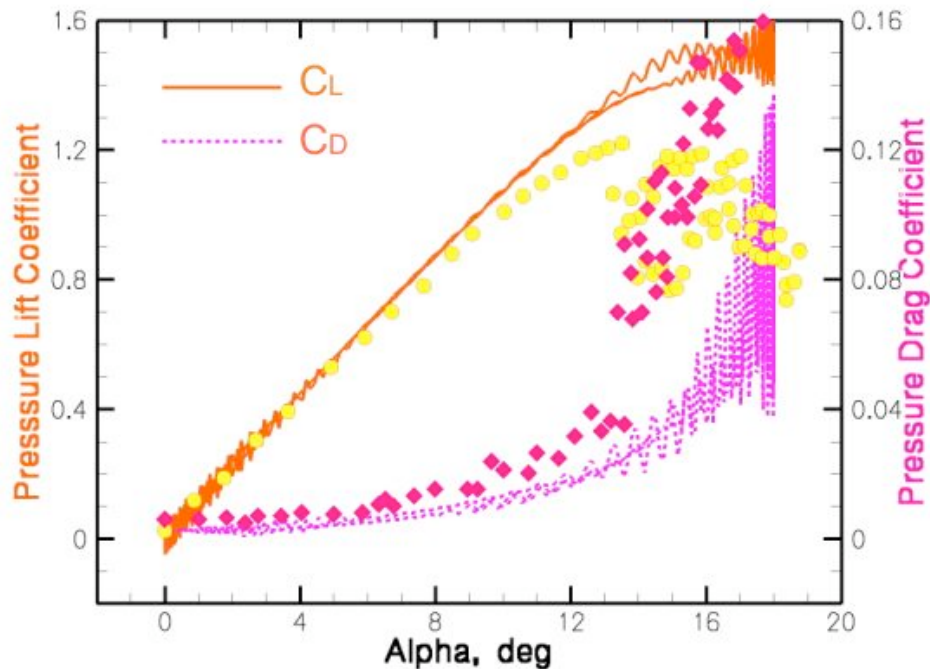
Oscillating Airfoil: Lift and Pitching Moment



This preliminary example is to demonstrate the unsteady behavior of CFD in the stalled regime. An intriguing feature is the repeatability of the large amplitude oscillation in lift and drag as we repeated the time cycles. The lift and drag data from the experiment did not come from a balance but instead was integrated values from an array of pressure transducers. Assuming that the suction peak on the airfoil may not be perfectly captured by the fixed position pressure transducers, the lift would be lower and the drag would be higher than their respective actual values.



α - Sweep through stall region





What Have We Learned

- Successfully simulated oscillating 2D airfoil and 3D semi-span wing and showed good agreement with wind tunnel data
- Demonstrated unsteady flow simulation for periodic pitch, yaw, and roll motions of a generic wing at a high subsonic Mach number
- Demonstrated the potential for static S&C database population by continuous sweep in α and β with an efficiency equal to or better than using steady state CFD
- High density S&C data computed without additional cost
- Stall region solutions showed very significant nonlinear effects which are difficult to capture in point-wise steady state CFD calculations



Future Work

- Validate time-dependent method for complex configurations and improve efficiency
- Develop close loop moving grid capabilities coupled to 6-DOF motion and structural deformation
- Use LES/DES/PANS codes to enhance accuracy for separated flow and vortex interaction
- Implement similar grid motion algorithm for unstructured mesh



cobalt solutions

Application of CFD to Abrupt Wing Stall Using RANS and DES

Jim Forsythe

Cobalt Solutions, LLC

NASA LaRC SAMS Contract NAS1-00135

Charles Fremaux, Robert Hall

**Acknowledgements: Joe Chambers, Paresh Parikh,
Scott Morton (USAF), ASC MSRC**

Outline

- Motivation
- Grid
- Static cases
 - Solution procedure
 - Results
- Oscillating roll cases
 - Solution procedure
 - Results
- Conclusions

Previous work published at AIAA meeting in Reno 2003, and to be published in AIAA Journal of aircraft (FOM=Figure of Merit).



Motivation?



- Pre-production F/A-18E
 - Exhibited “wing drop” in flight test
 - “wing drop” is an uncommanded lateral motion
 - “abrupt wing stall” is an aerodynamic characteristic, and can cause wing drop
 - Numerous flight tests resulting in a production fix
 - Revised flight control laws and porous wing fold fairing
 - A comprehensive program was created to be able to predict these phenomenon with wind tunnels and CFD
 - Free-to-roll wind tunnel test method
 - FOM’s for steady and unsteady (non-moving) CFD
- **Current work: Progress CFD to calculations of damping derivatives and free-to-roll for this flow by application to pre-production F/A-18E**

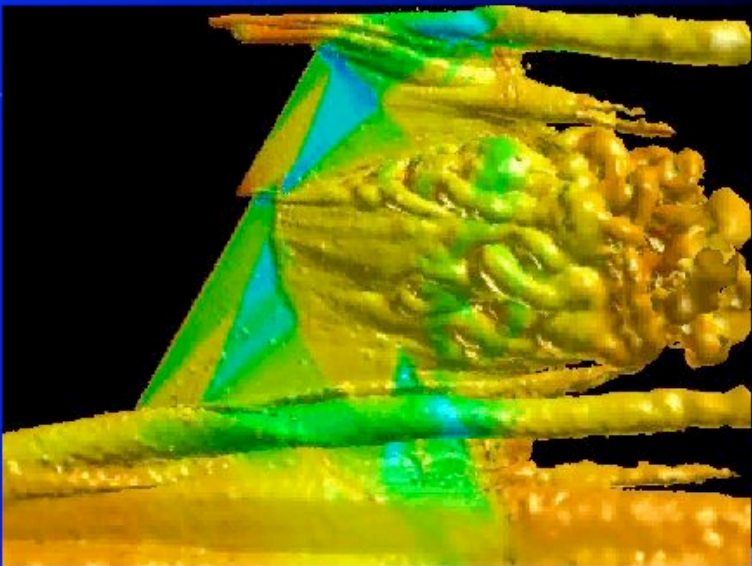
For DES, RANS is responsible for predicting boundary layer growth and separation. LES is responsible for predicting the geometry dependant turbulent flow features. Grid adaptation done using NASA Langley's RefineMesh program. Adaptation on time average of vorticity

coherent solutions

Previous work (AIAA 03-0594)

- Goal was to use CFD to predict the unsteady shock oscillations seen in the experiments.
- RANS models failed to give unsteady results
- Detached-Eddy Simulation turbulence model
 - Hybrid RANS/LES
 - RANS in boundary layer
 - LES outside of boundary layer
- Solution based grid adaption

Iso of vorticity, colored by pressure, $\alpha=9^\circ$

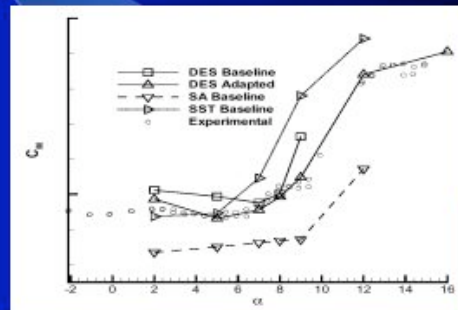
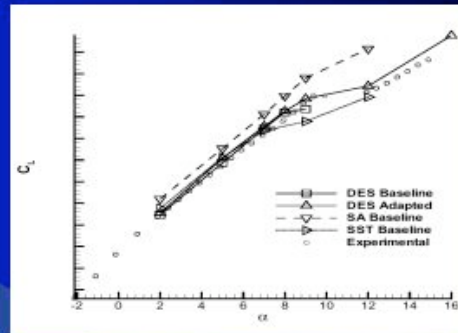


DES results are time-averaged coefficients. Left axis removed to protect proprietary data.

coalt solutions

Previous work (AIAA 03-0594)

- 10/10/5 flap set with no tails
- SST predicted early lift curve break
- DES showed an improved lift curve break (but on a grid finer than the current grid)
- Motivates inclusion of DES in the current project, along with RANS

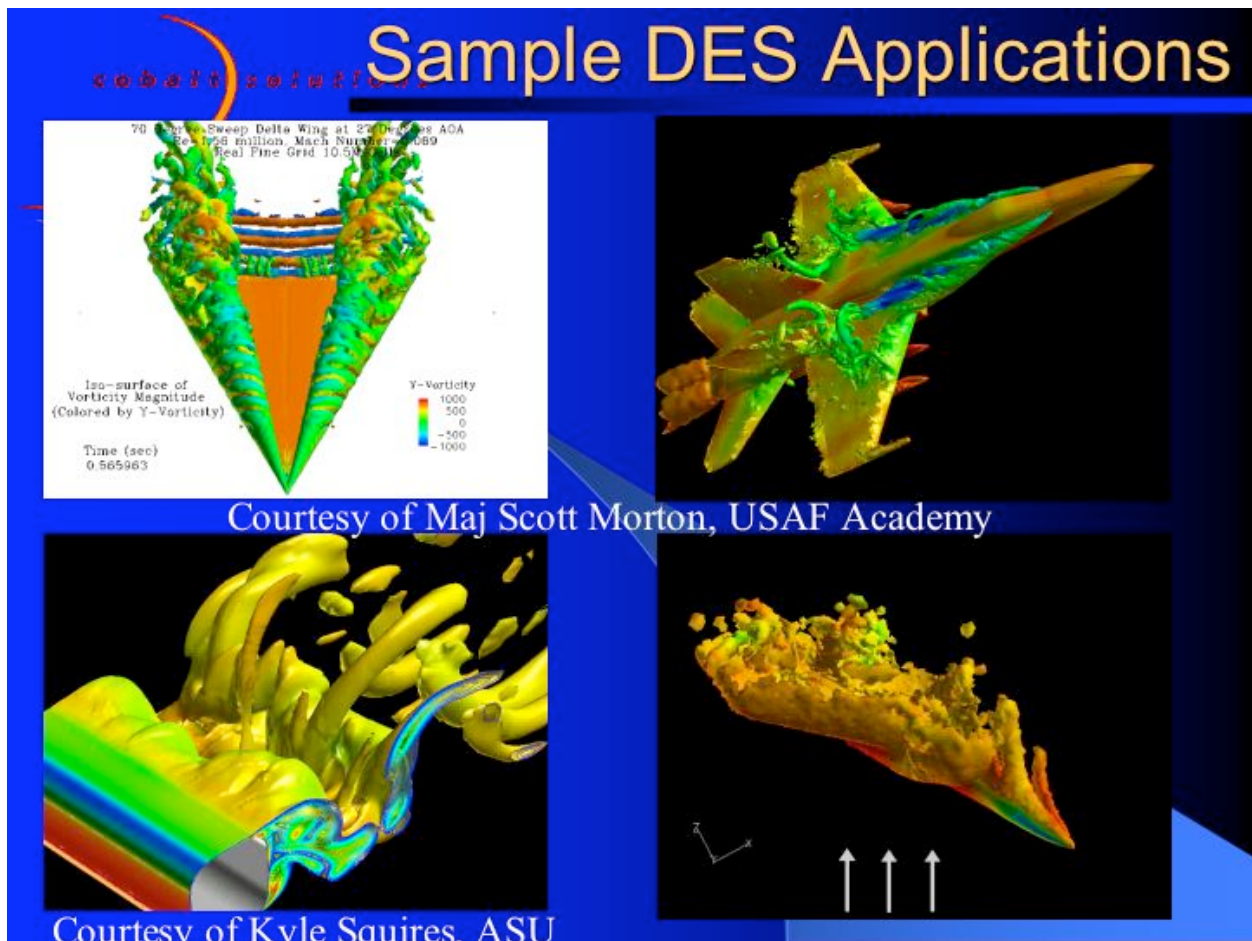


These DES projects represent a cross section of those done over the past few years using Cobalt.

Delta wing vortex breakdown on a delta wing and the F-18C done by Major Scott Morton of the USAF Academy (Scott.morton@usafa.af.mil).

2-D forebody geometry by Kyle Squires (squires@asu.edu).

Prescribed spin of the F-15E by James Forsythe.

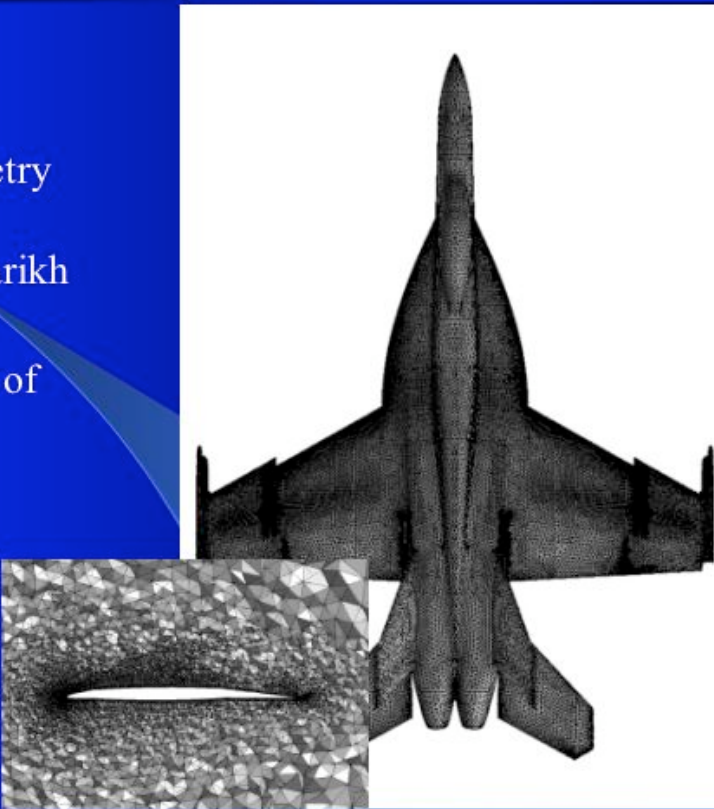


Prisms created using “Blacksmith” to recombine the tets in the boundary layer into prisms. Blacksmith is a Cobalt grid utility.

cobalt solutions

Grid

- Grid mirrored about symmetry plane
- Grid provided by Paresh Parikh
- 6/8/4 flap set
- 8.4×10^6 cells for both sides of aircraft
- Adaption performed on a 9° time-averaged DES solution under previous work
- Prisms in boundary layer
- Average $y^+ < 0.7$

The image is a composite. The main part is a top-down view of a fighter jet, likely an F-16, with a dense, dark grid of cells covering its entire surface and the surrounding flow field. The jet is centered vertically. In the bottom-left corner, there is a smaller, square inset showing a close-up, perspective view of the grid cells near a surface, highlighting the transition from tetrahedra to prisms in the boundary layer.

The following are non-moving cases – but can be unsteady (for DES)



CPU hours based on a Compaq ES45. Timestep for DES non-dimensionalized by chord and freestream velocity.



Solution Procedure

- Menter's SST RANS model
 - Convergence monitored by observing forces and moments. Rolling moment was generally the most sensitive and last to converge.
 - 4000 iterations
 - 1 Newton sub-iteration
 - CFL of 1×10^6
 - 2000 cpu hours per run.
- Spalart-Allmaras based DES model
 - Unsteady flow simulation
 - 16000 iterations
 - 3 Newton sub-iterations
 - $\Delta t^* = 0.01$
 - 8x the cost of the steady RANS simulations

Model was set to a given pitch angle (theta), then rolled about the longitudinal axis (phi). This resulted in a decrease of alpha, and an increase in beta as phi increased. The CFD was performed at the given alphas and betas, which were corrected in the wind tunnel data for wall effects.



Test Matrix

- Conditions chosen to match NASA Langley wind tunnel test 565
- Mach=0.9
- $Re_c = 3.9 \times 10^6$
- Flow through engines
- Sting not included in grid

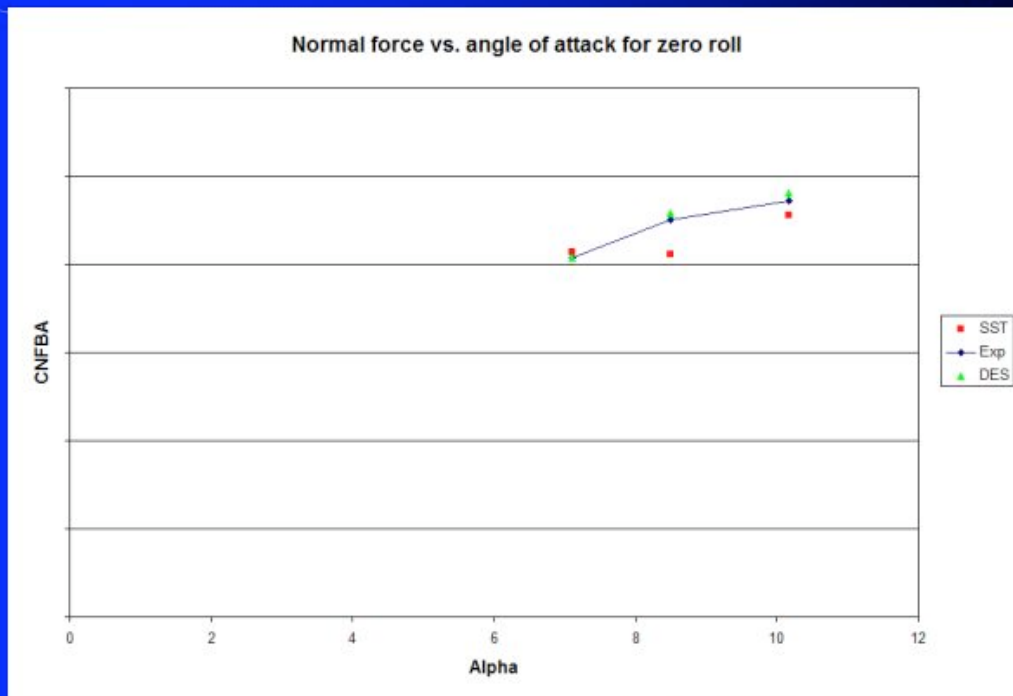
247			
ALPHA	BETA	PHI	THETA
7.11	-0.14	-0.13	7.01
7.03	0.86	9.87	6.97
6.32	2.73	29.75	6.74
4.03	4.89	59.78	6.14

240			
ALPHA	BETA	PHI	THETA
8.50	0.35	3.84	8.40
8.12	1.04	9.82	8.08
7.58	3.42	29.63	8.18

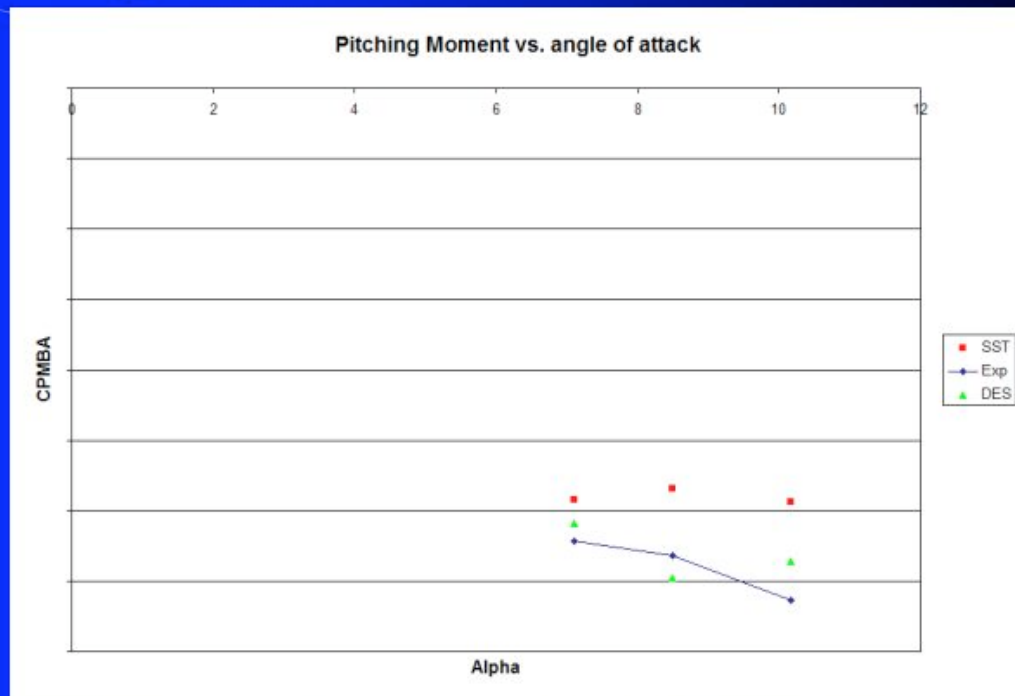
242			
ALPHA	BETA	PHI	THETA
9.04	1.17	9.69	9.00
8.20	3.74	29.56	8.87

244			
ALPHA	BETA	PHI	THETA
10.17	-0.12	-0.02	10.06
10.05	1.38	9.79	10.03
9.06	4.23	29.49	9.86
5.69	7.54	59.53	9.26

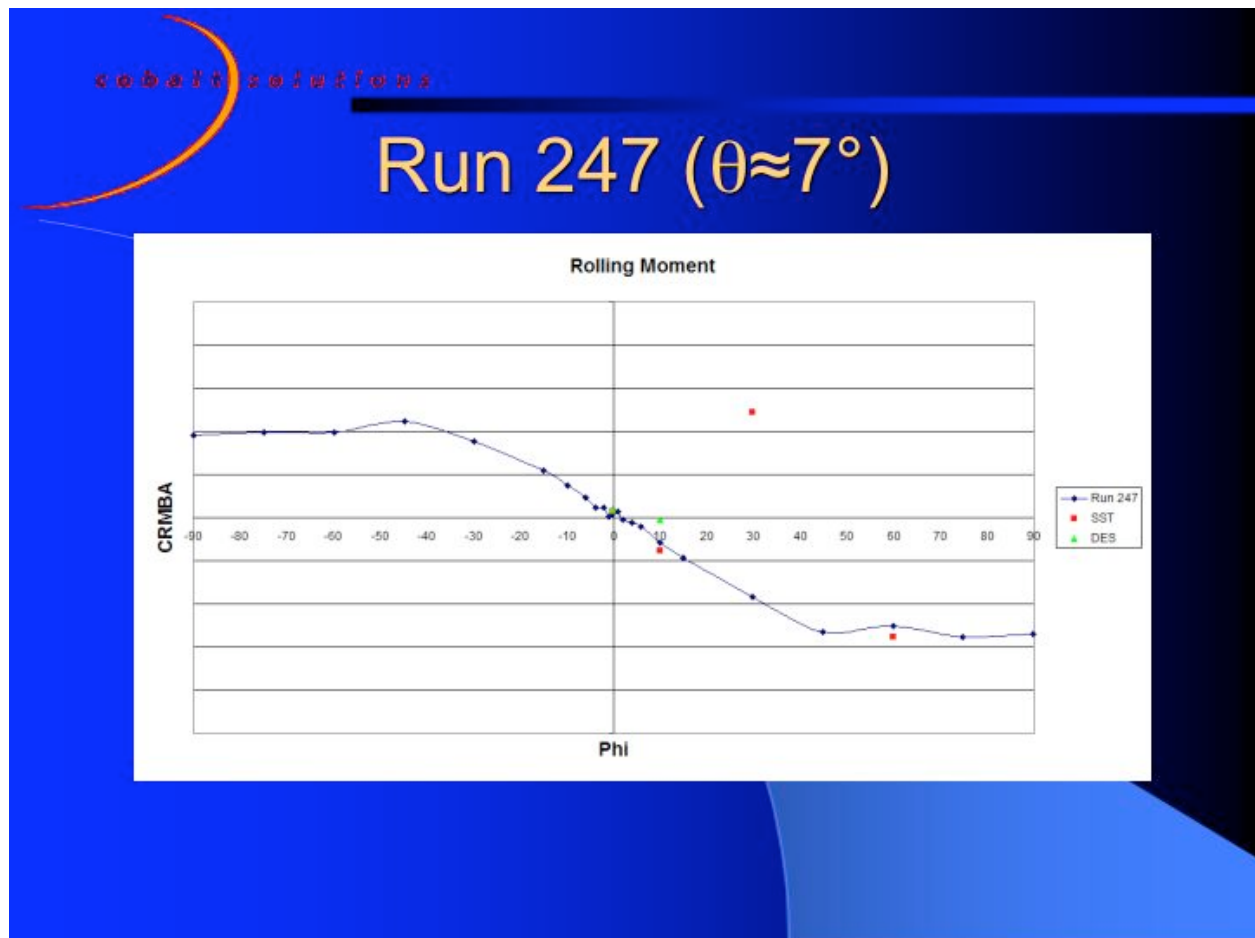
Normal force for near-zero sideslip



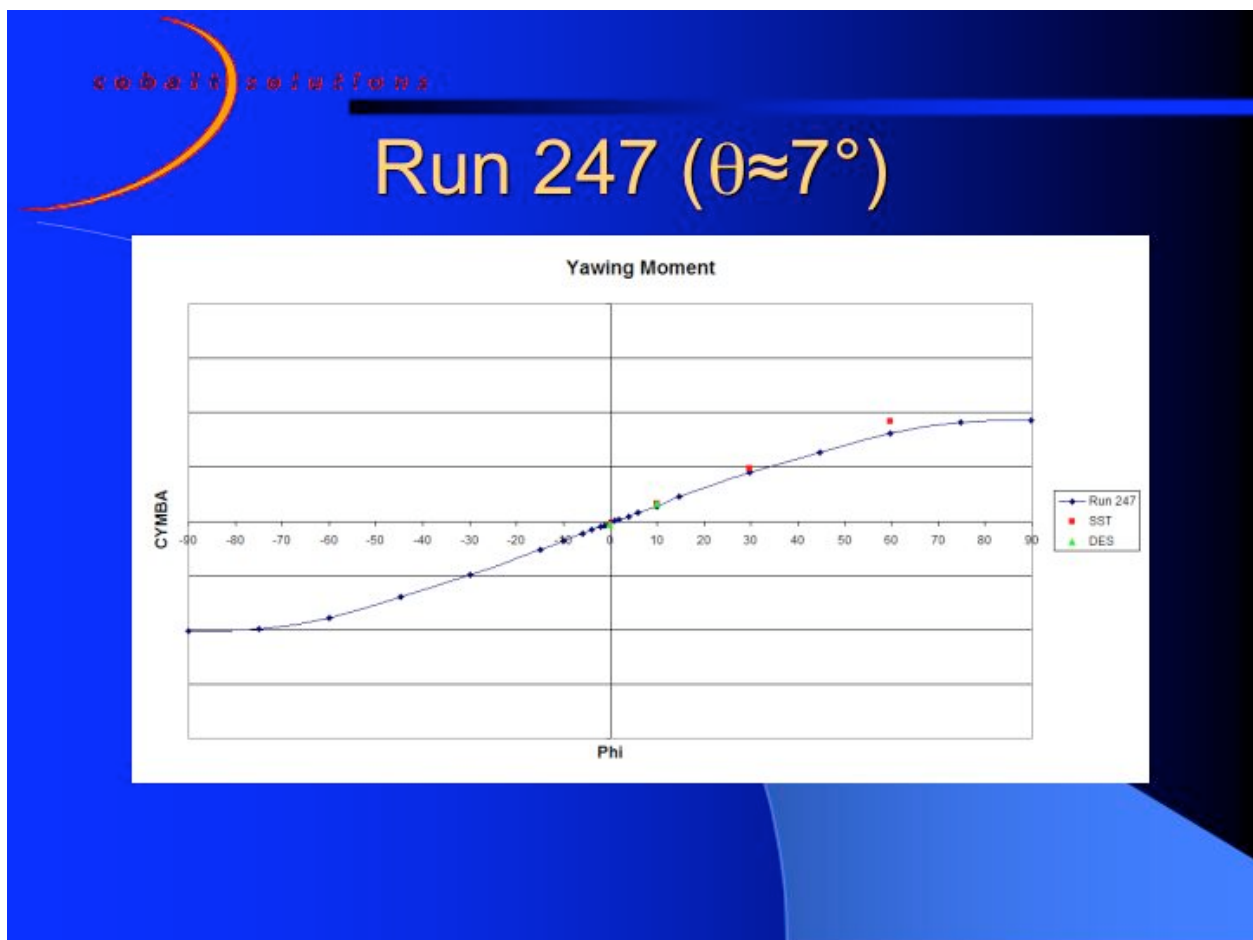
Pitching moment for near-zero sideslip



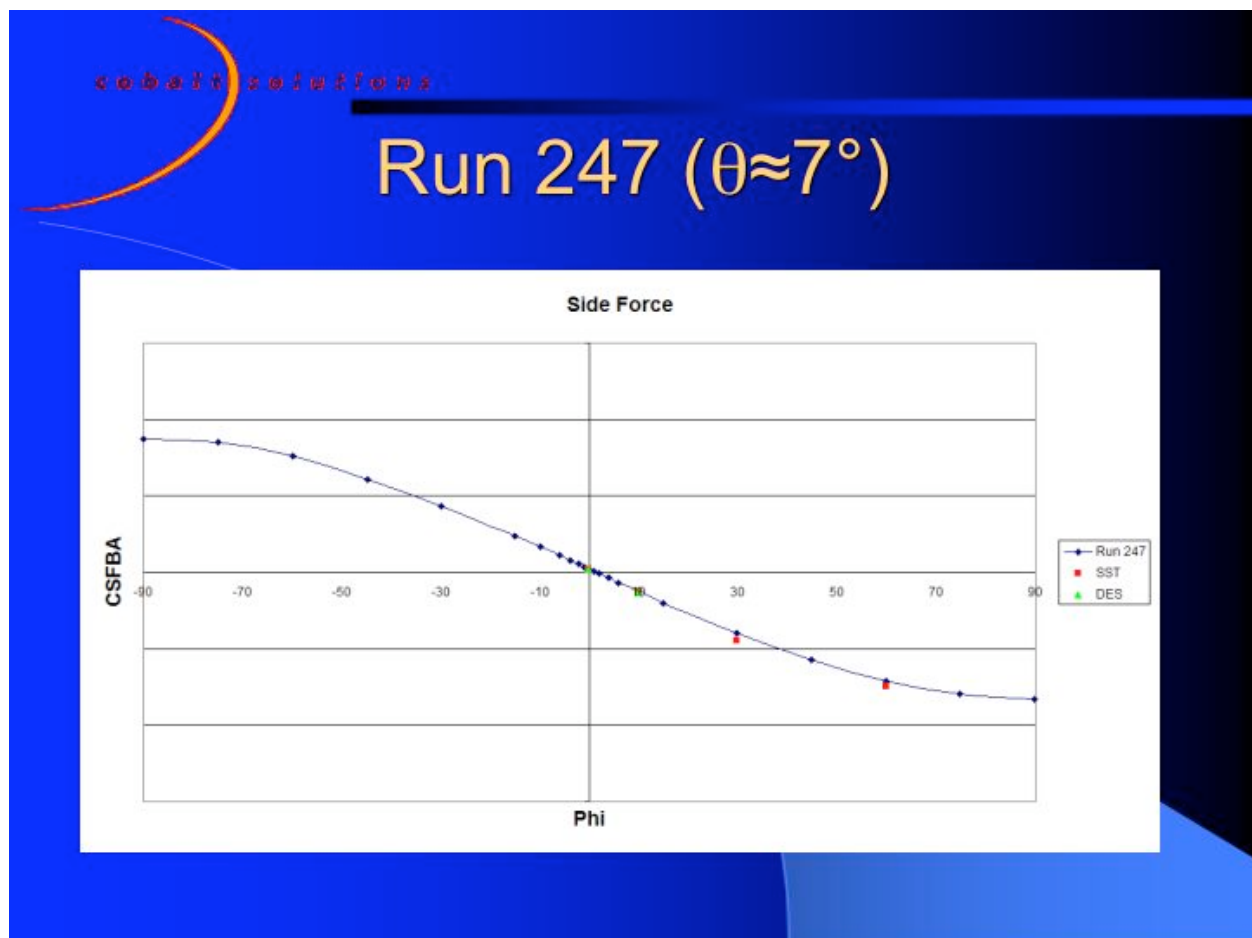
Note reversal of rolling moment for $\phi=30$ using SST.



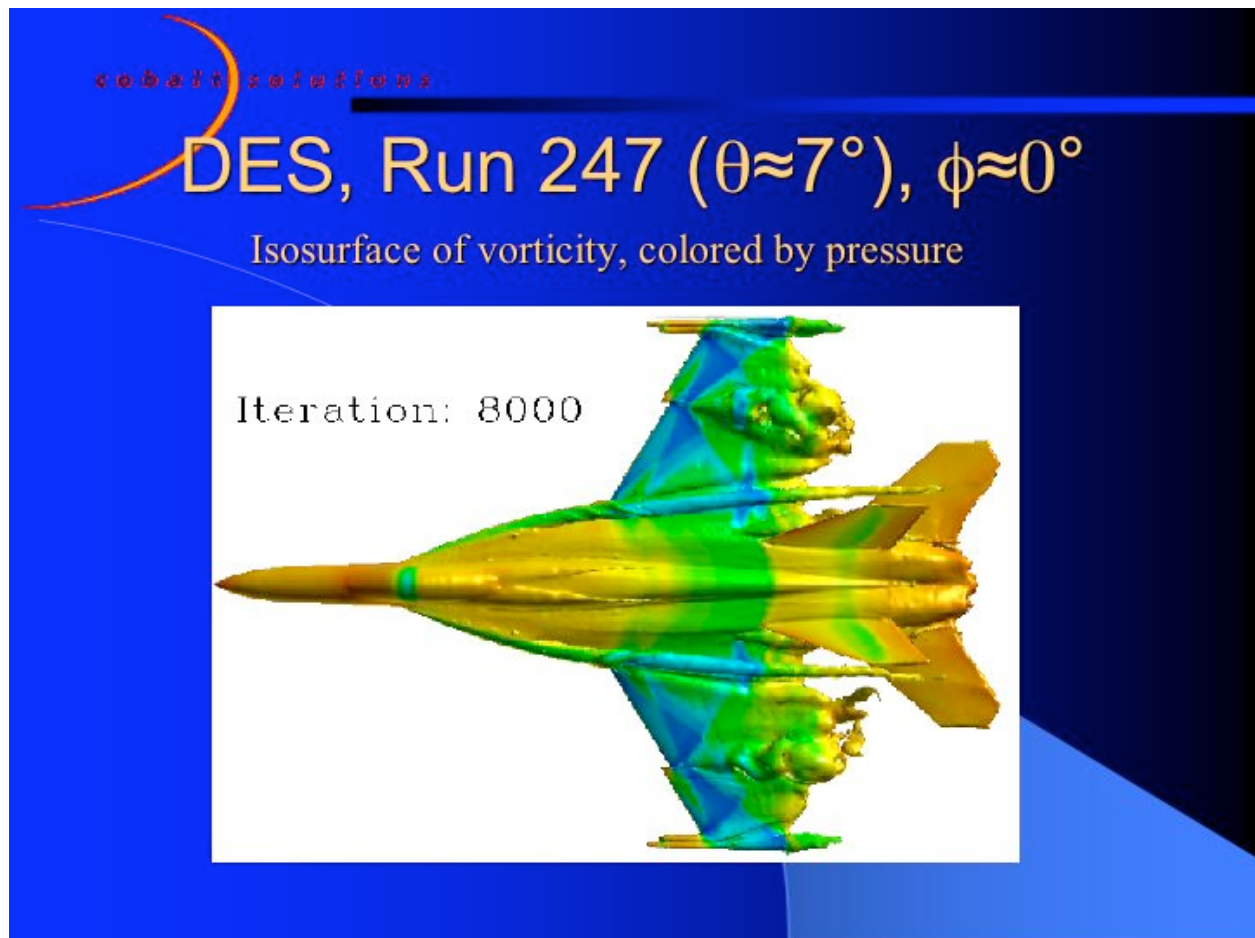
Yawing moment well predicted – as with all cases.



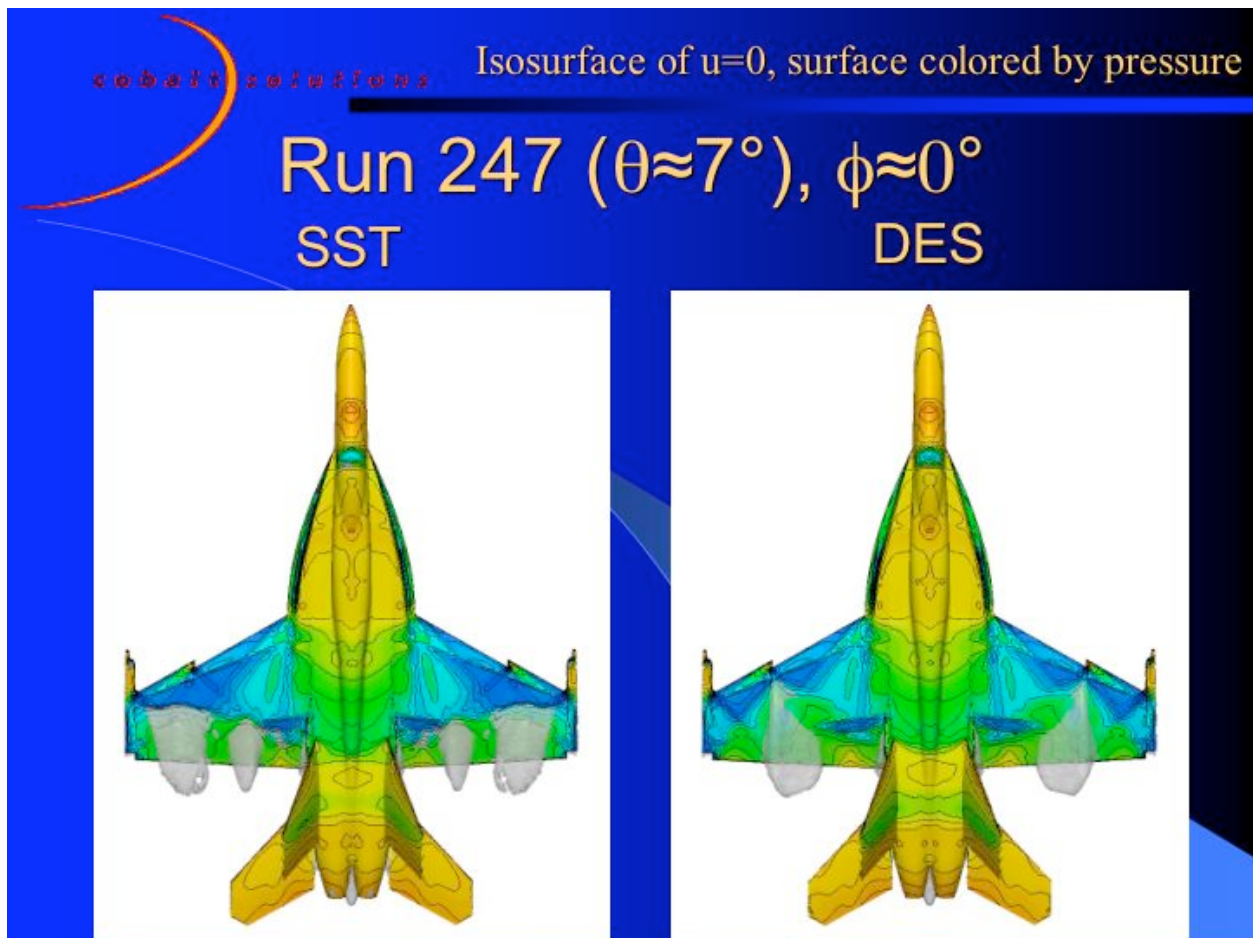
Side force well predicted – as with all cases.



Shock retreating off trailing edge of leading edge flap.



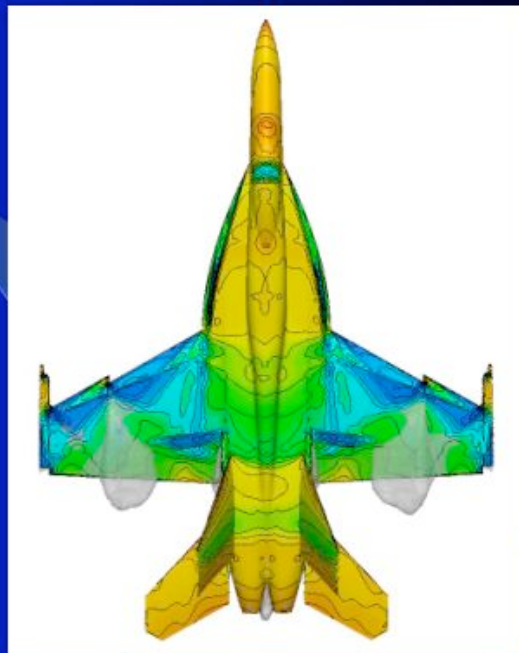
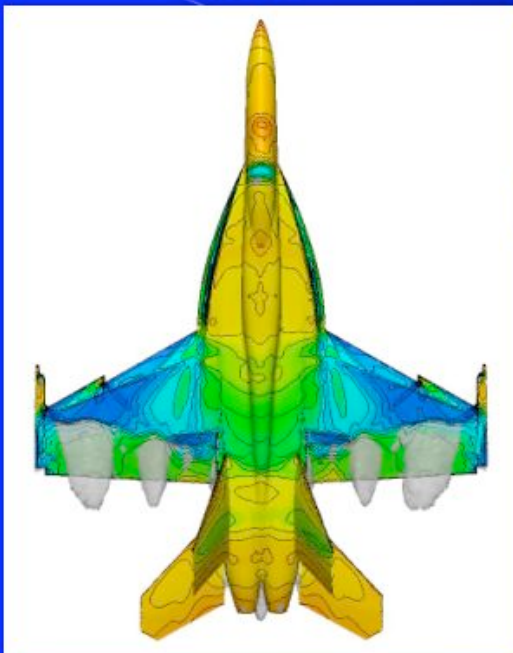
DES isosurface looks like separation is at trailing of leading edge flap. But it moves back from there unsteadily. This leads to the blue low pressure in the separation bubble (since it is not always separated).



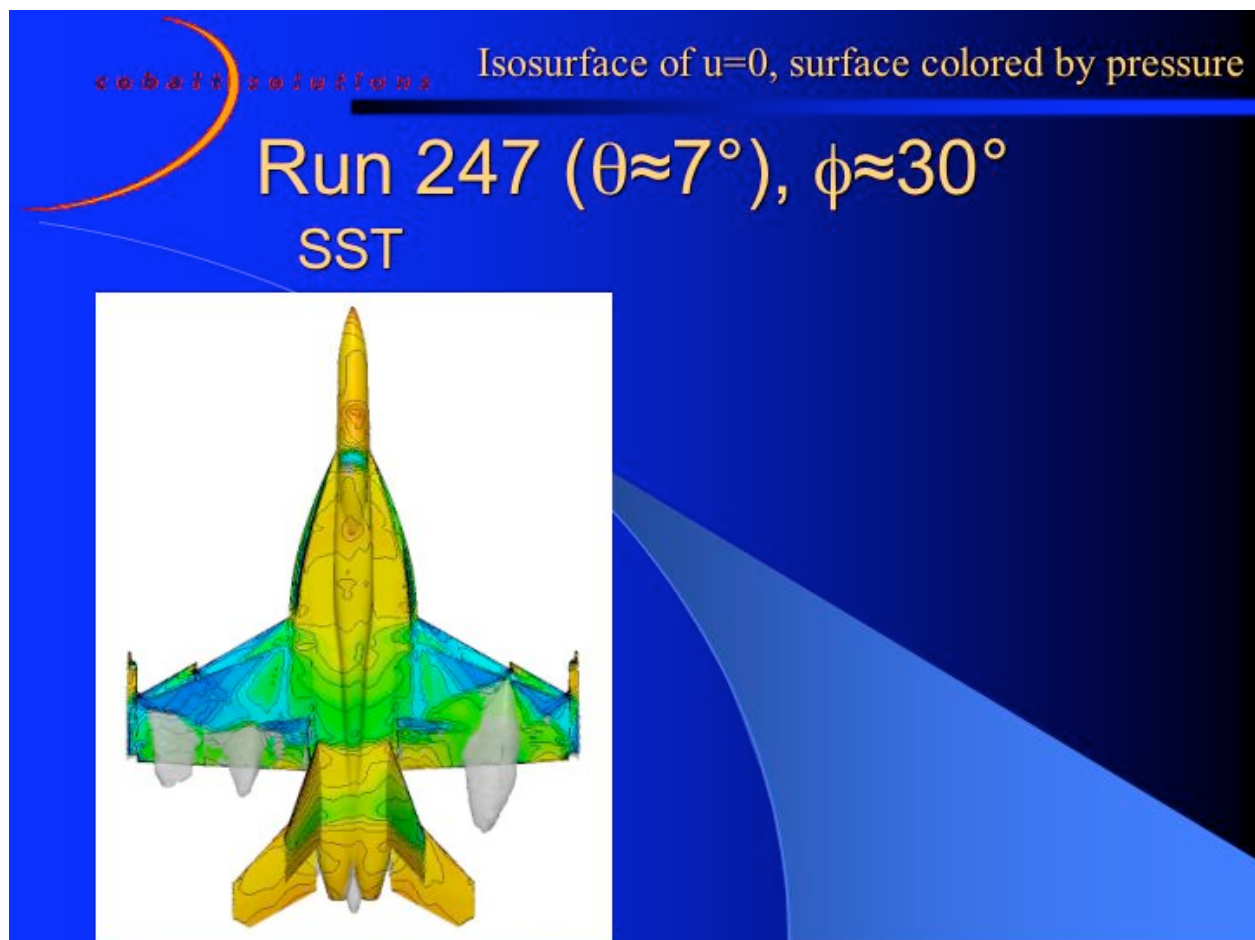
Run 247 ($\theta \approx 7^\circ$), $\phi \approx 10^\circ$

SST

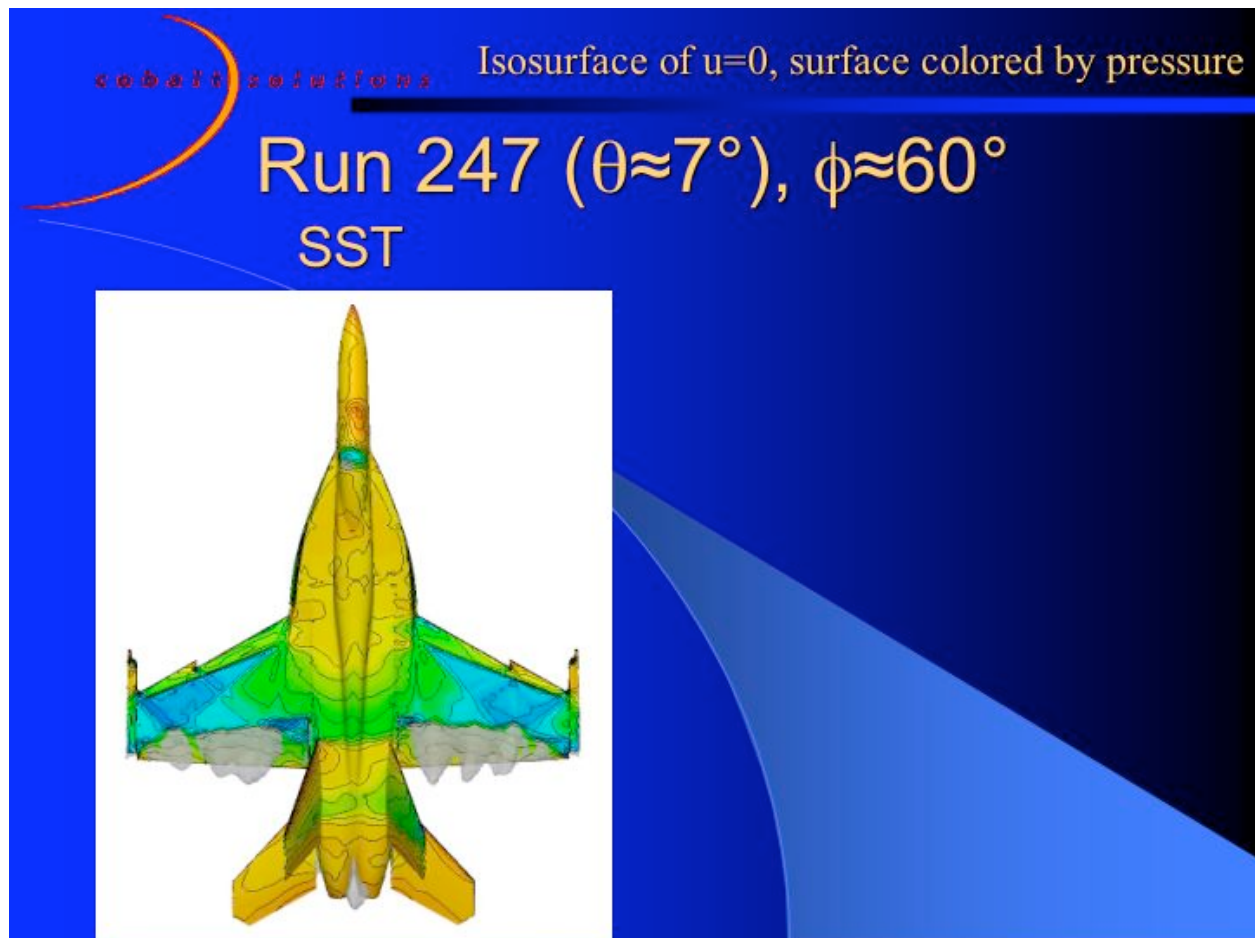
DES



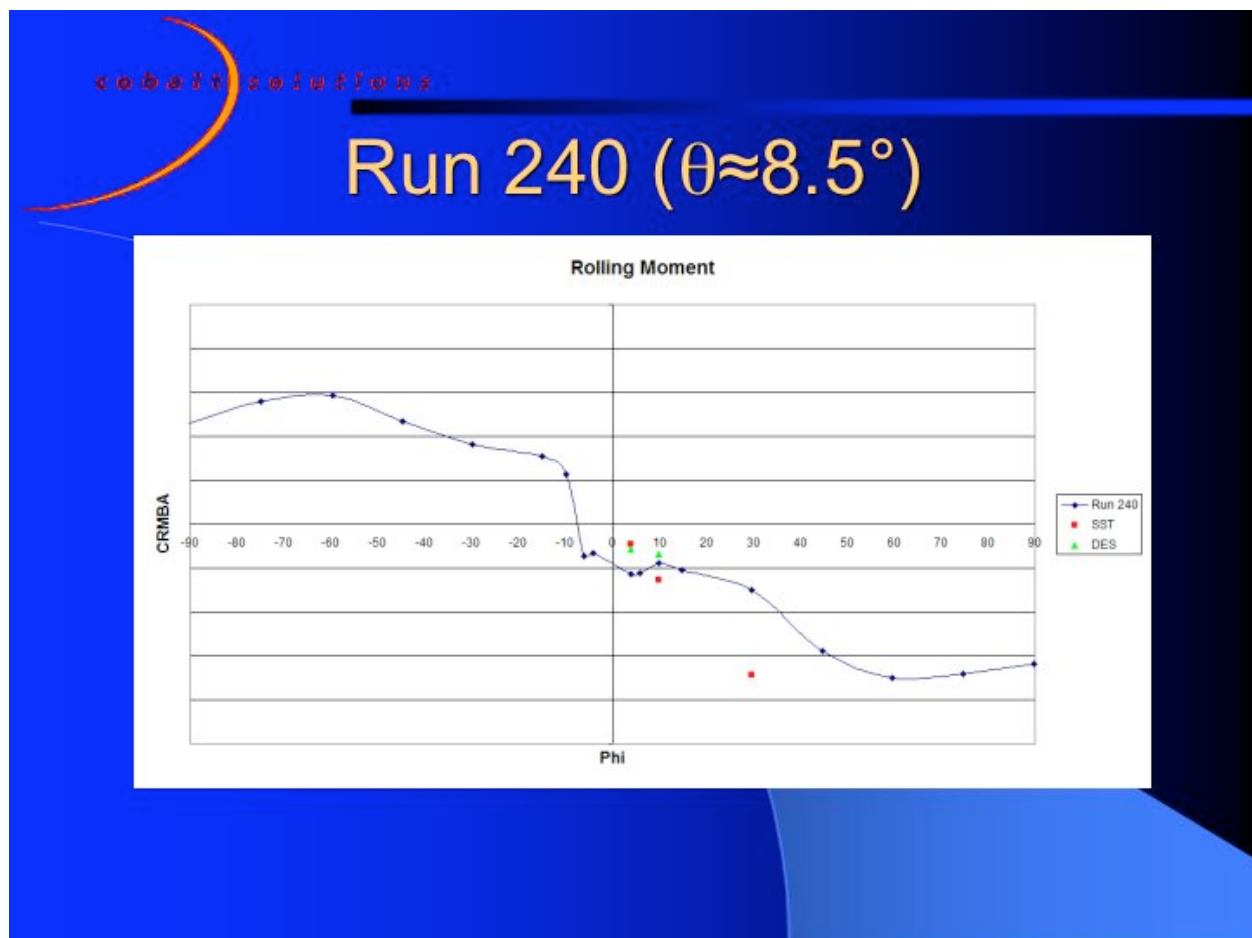
The separation moving forward on the right wing is the cause for the roll moment reversal.



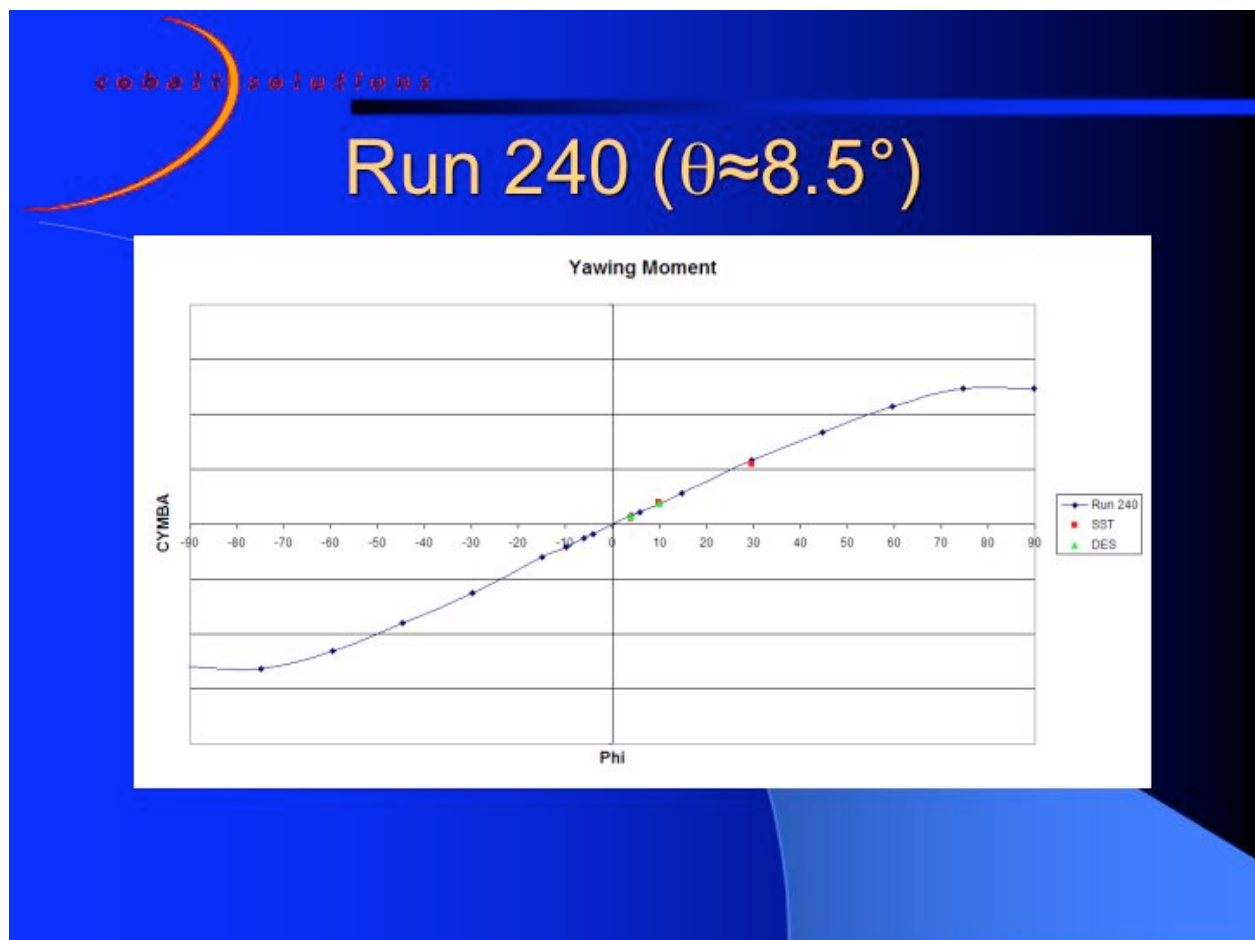
At this high ϕ , the α is reduced so much that the flow remains attached until the trailing edge of the wing.



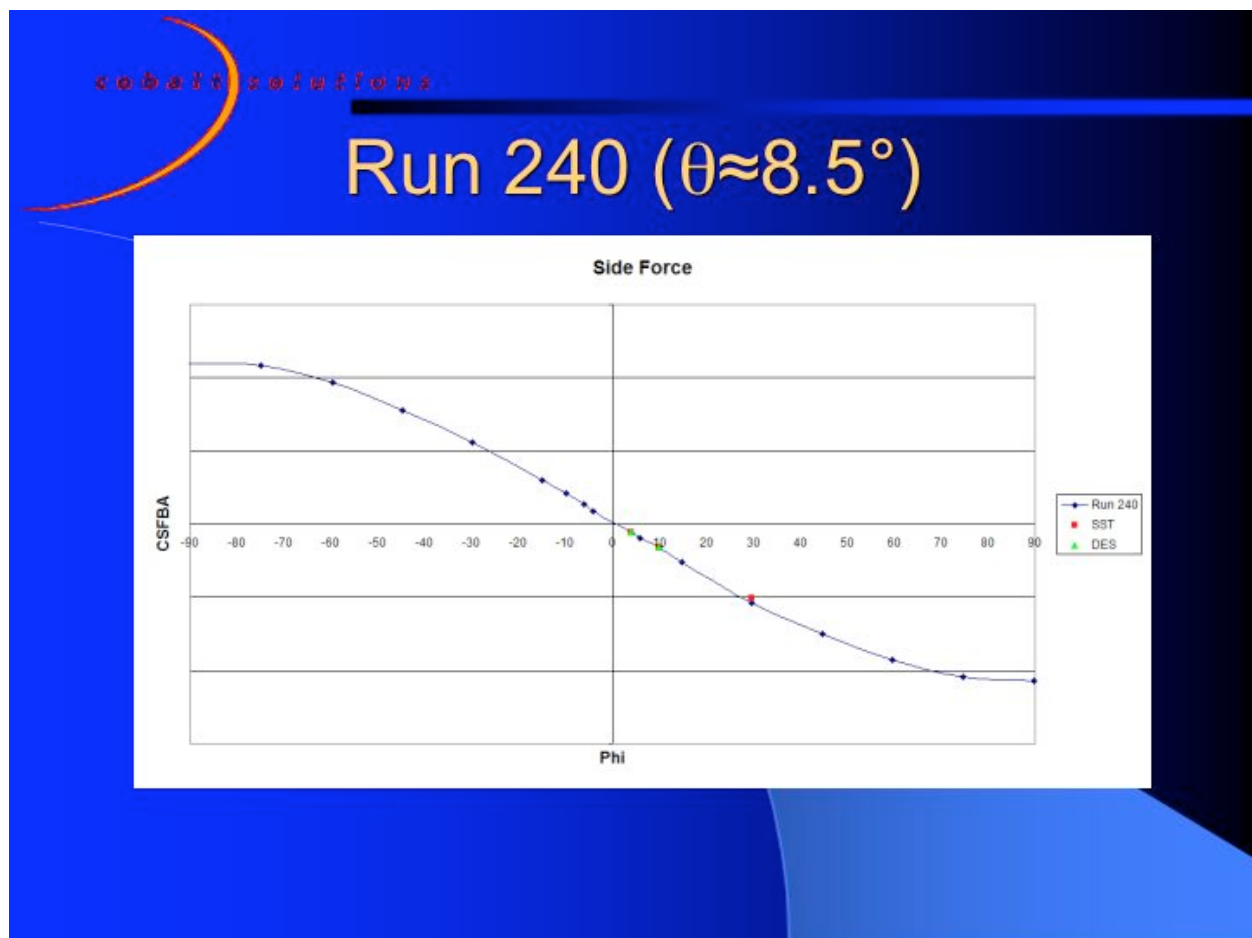
Note asymmetries in wind tunnel data. Decrease in lateral stability derivative picked up with DES.



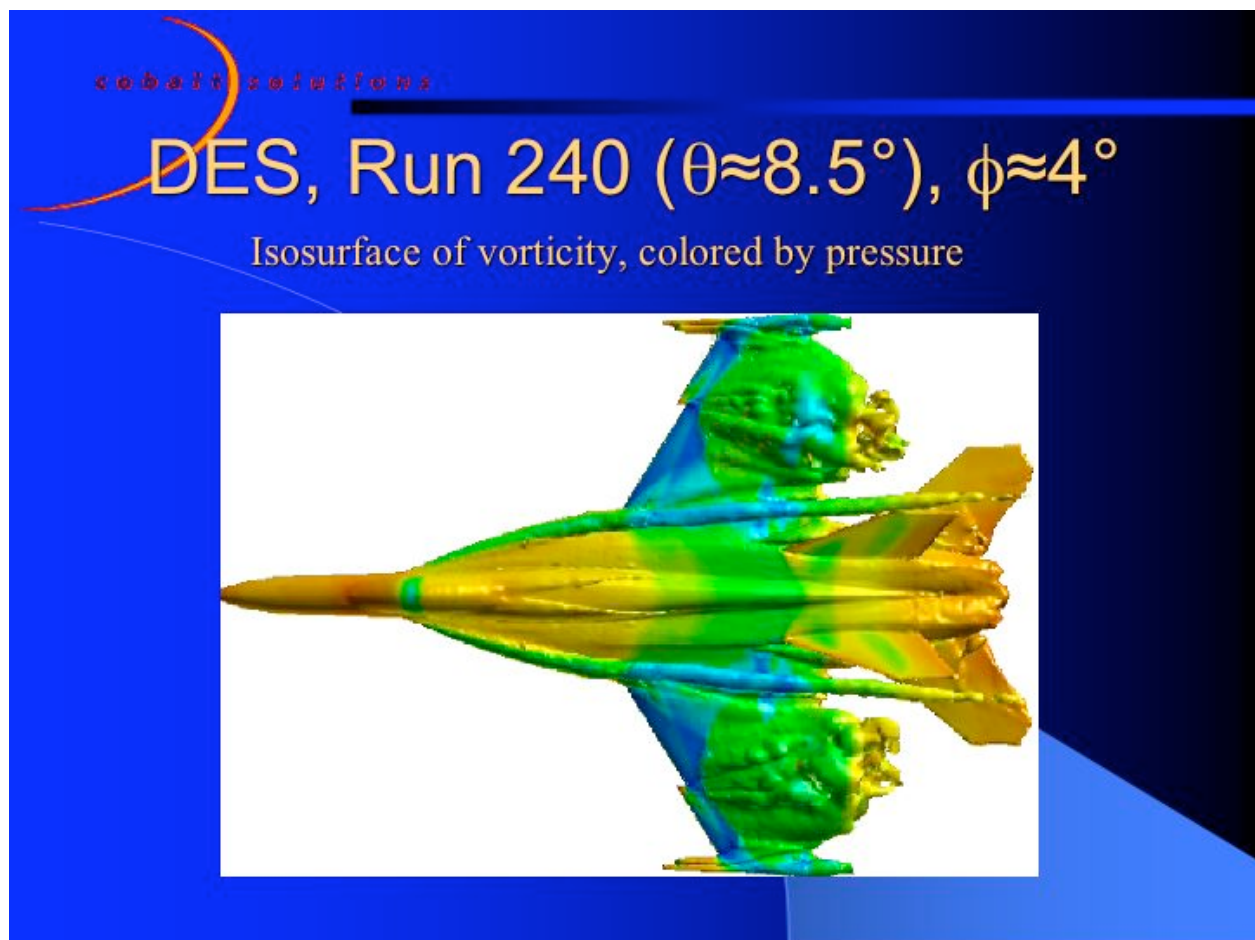
Good agreement for yawing moment, as with all cases – this is likely due to the attached flow at the tail, which is easily predicted.



Good agreement for side force, as with all cases – this is likely due to the attached flow at the tail, which is easily predicted.



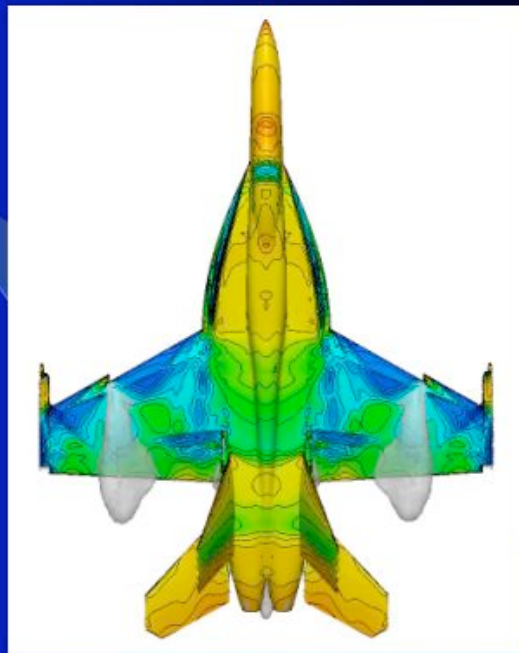
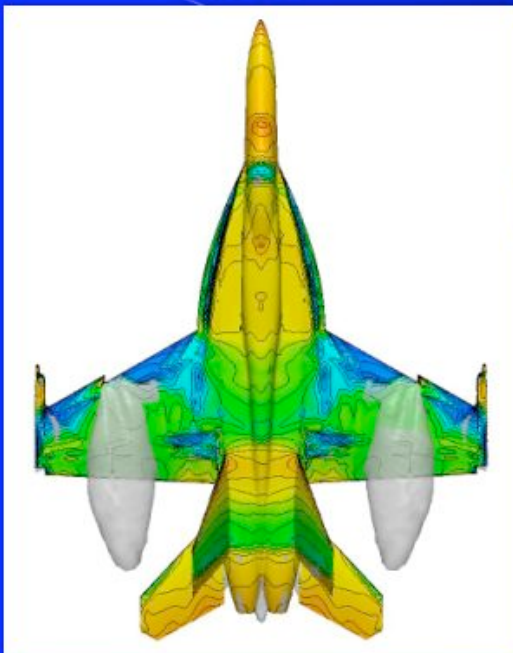
Separation is making it onto the leading edge of the leading edge flap.



Run 240 ($\theta \approx 8.5^\circ$), $\phi \approx 4^\circ$

SST

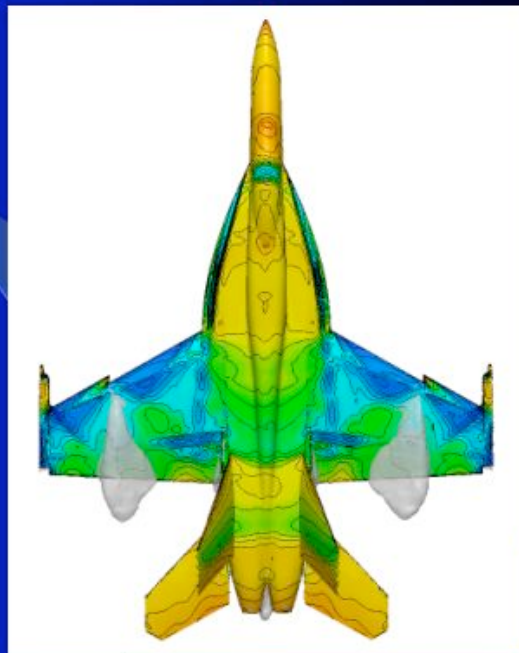
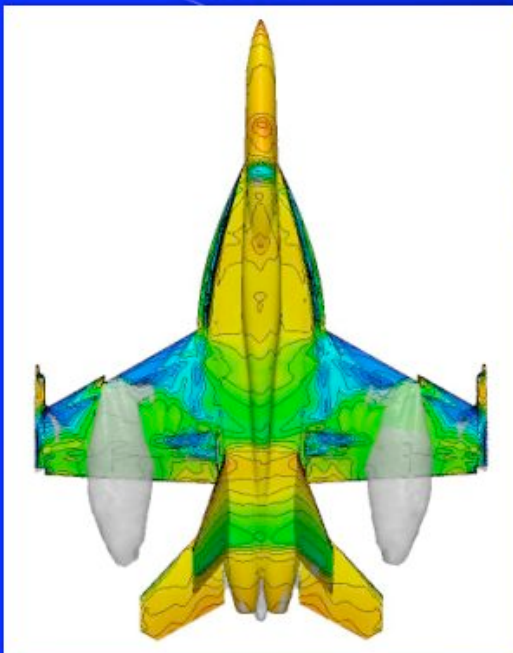
DES



Run 240 ($\theta \approx 8.5^\circ$), $\phi \approx 10^\circ$

SST

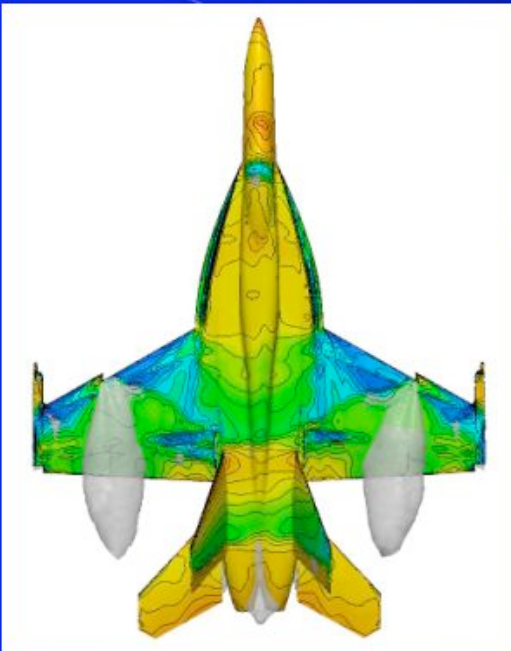
DES



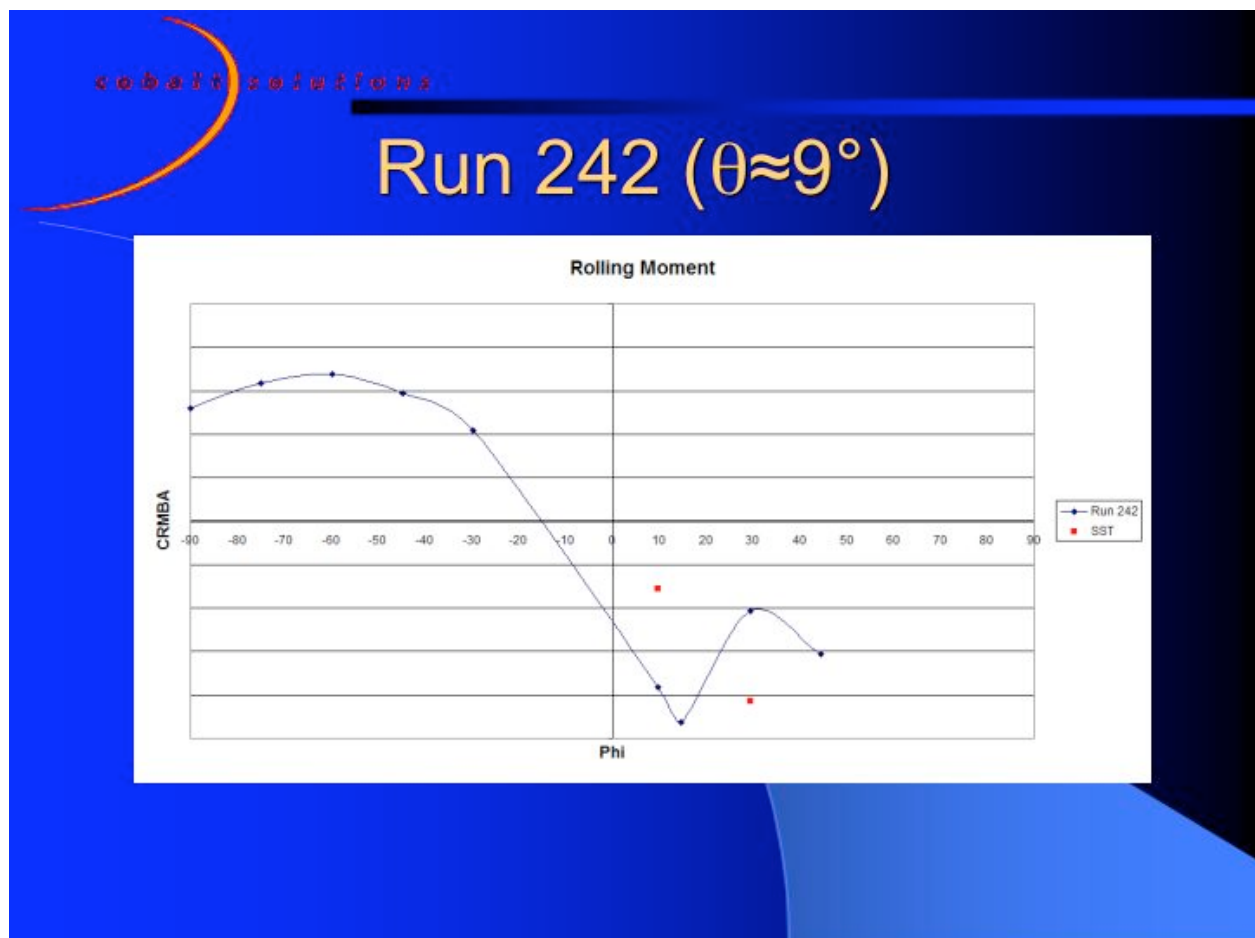
coherent solutions

Isosurface of $u=0$, surface colored by pressure

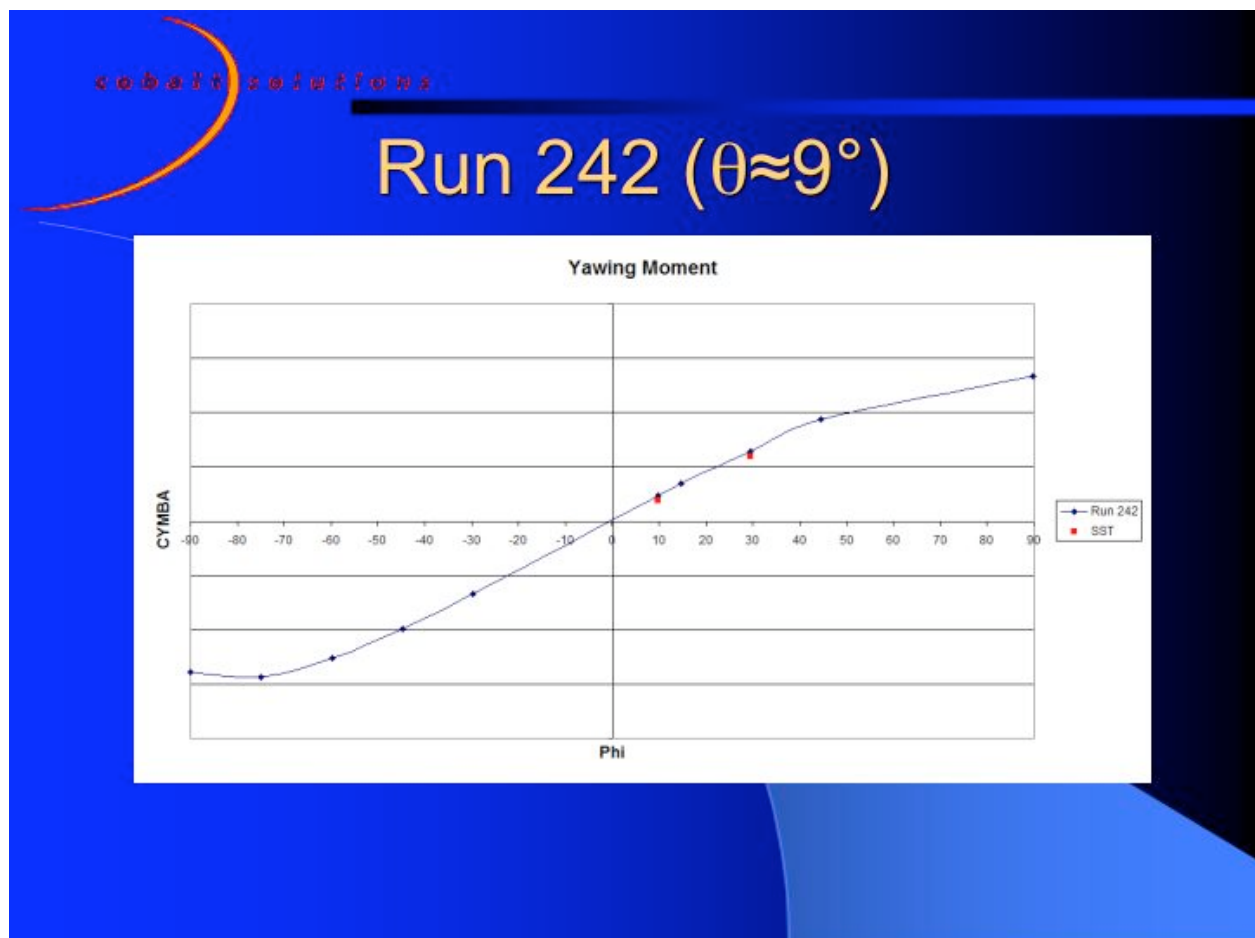
Run 240 ($\theta \approx 8.5^\circ$), $\phi \approx 30^\circ$
SST



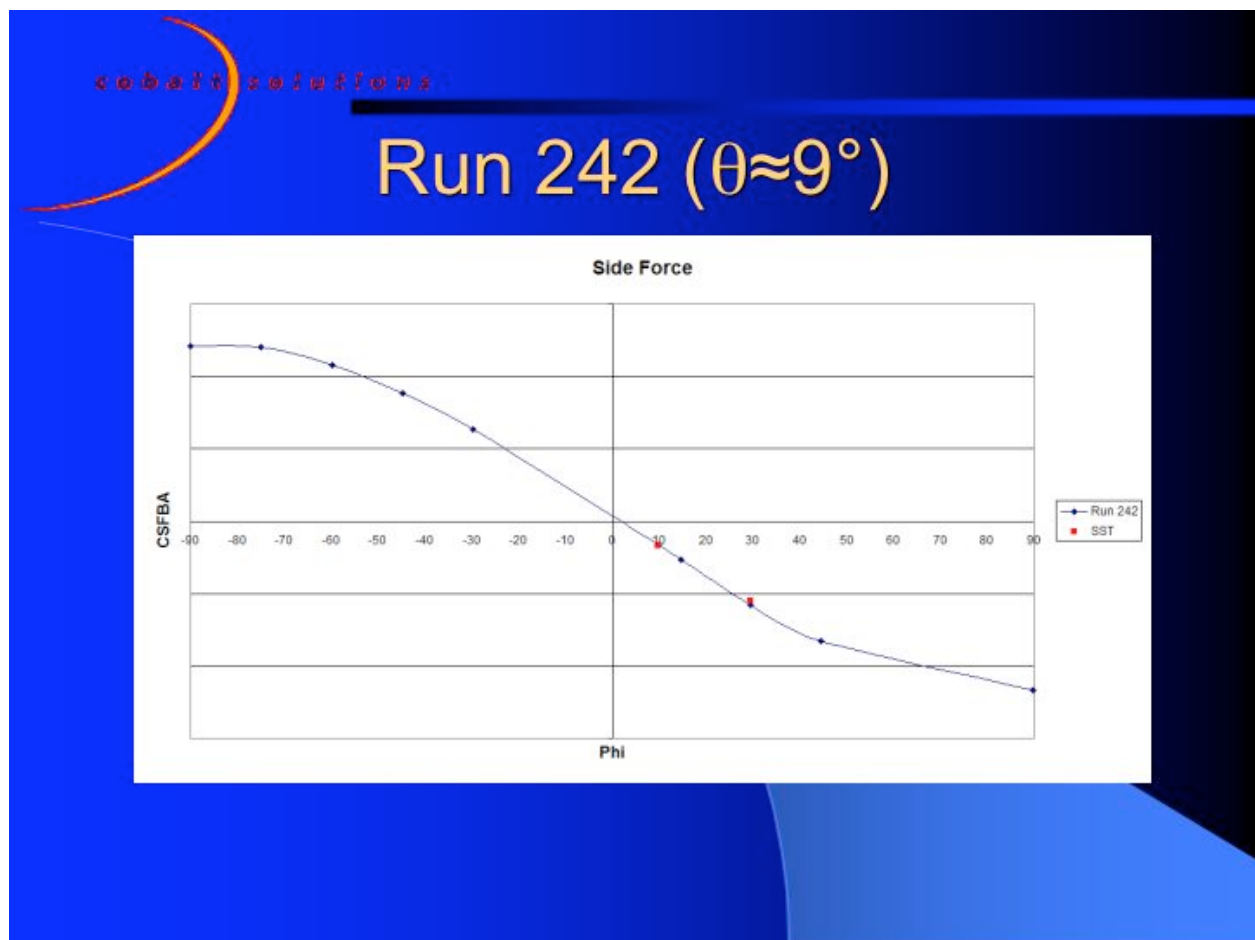
Large asymmetries in wind tunnel data. Around this angle there was difficulty in testing, since model dynamics became significant.



Good agreement for yawing moment, as with all cases.



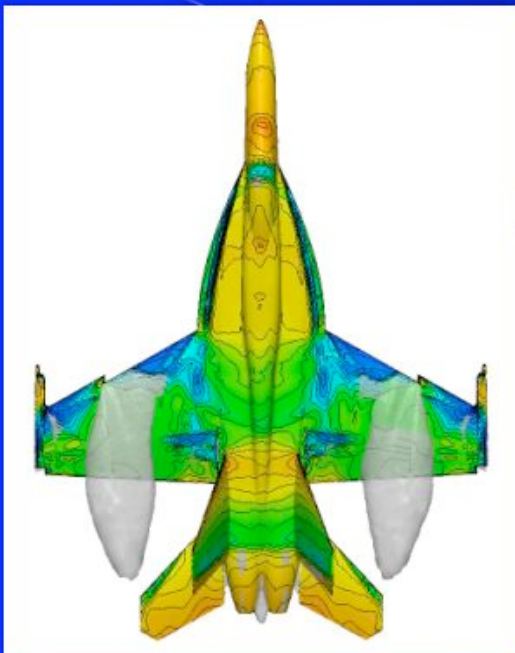
Good agreement for side force, as with all cases.



ceba solutions

Isosurface of $u=0$, surface colored by pressure

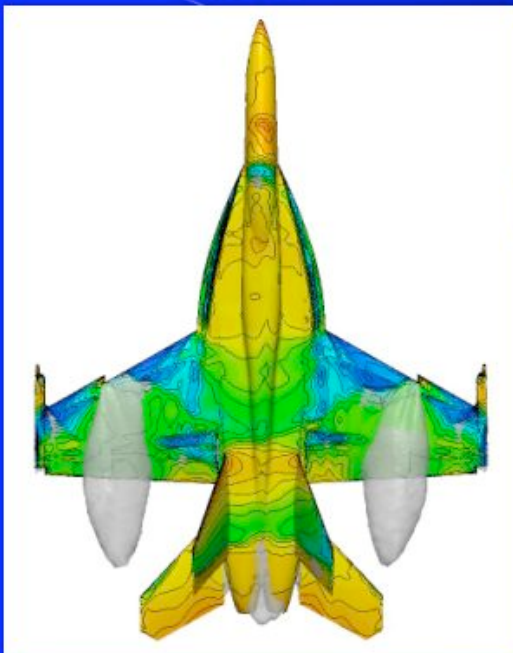
Run 242 ($\theta \approx 9^\circ$), $\phi \approx 10^\circ$
SST



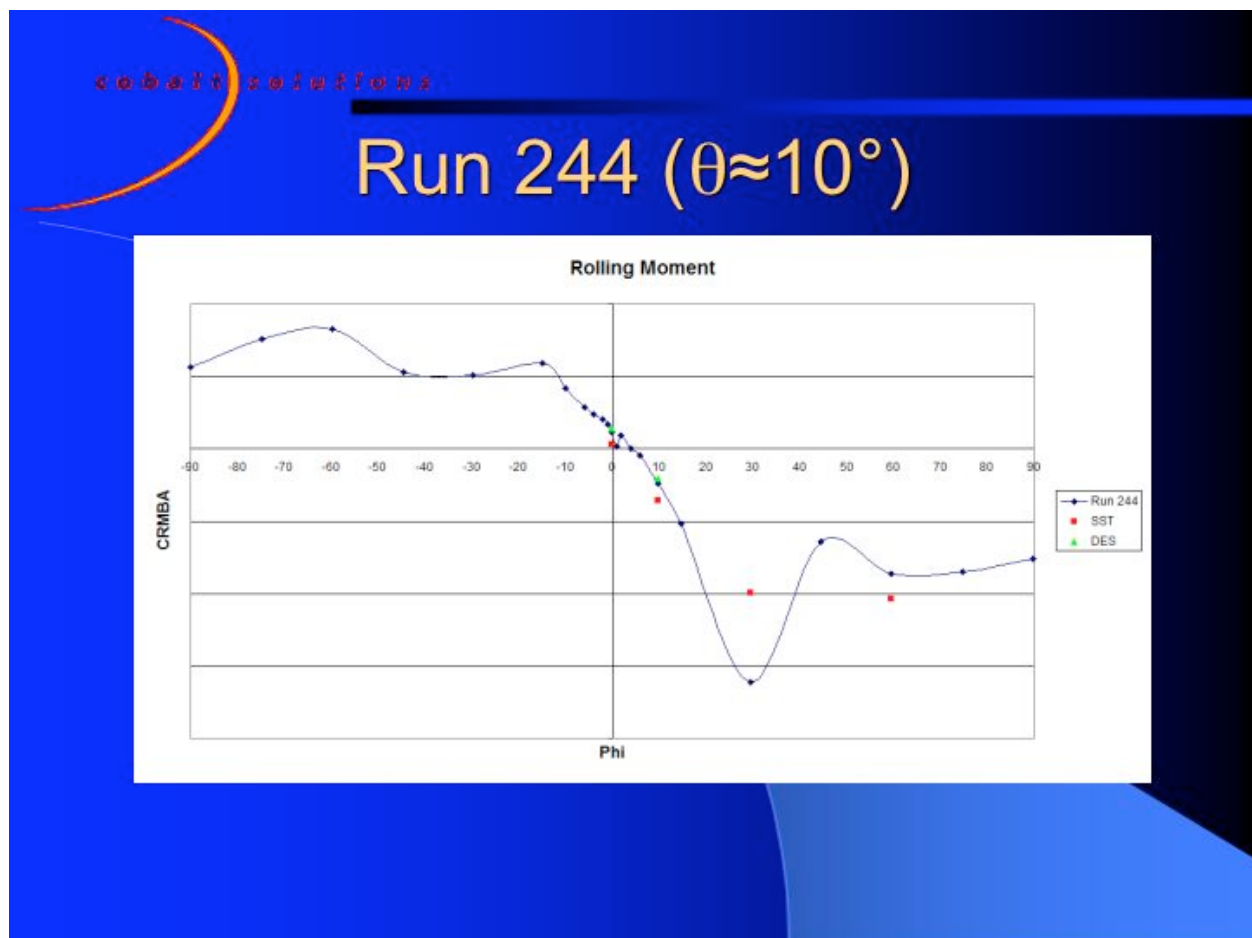
ceba solutions

Isosurface of $u=0$, surface colored by pressure

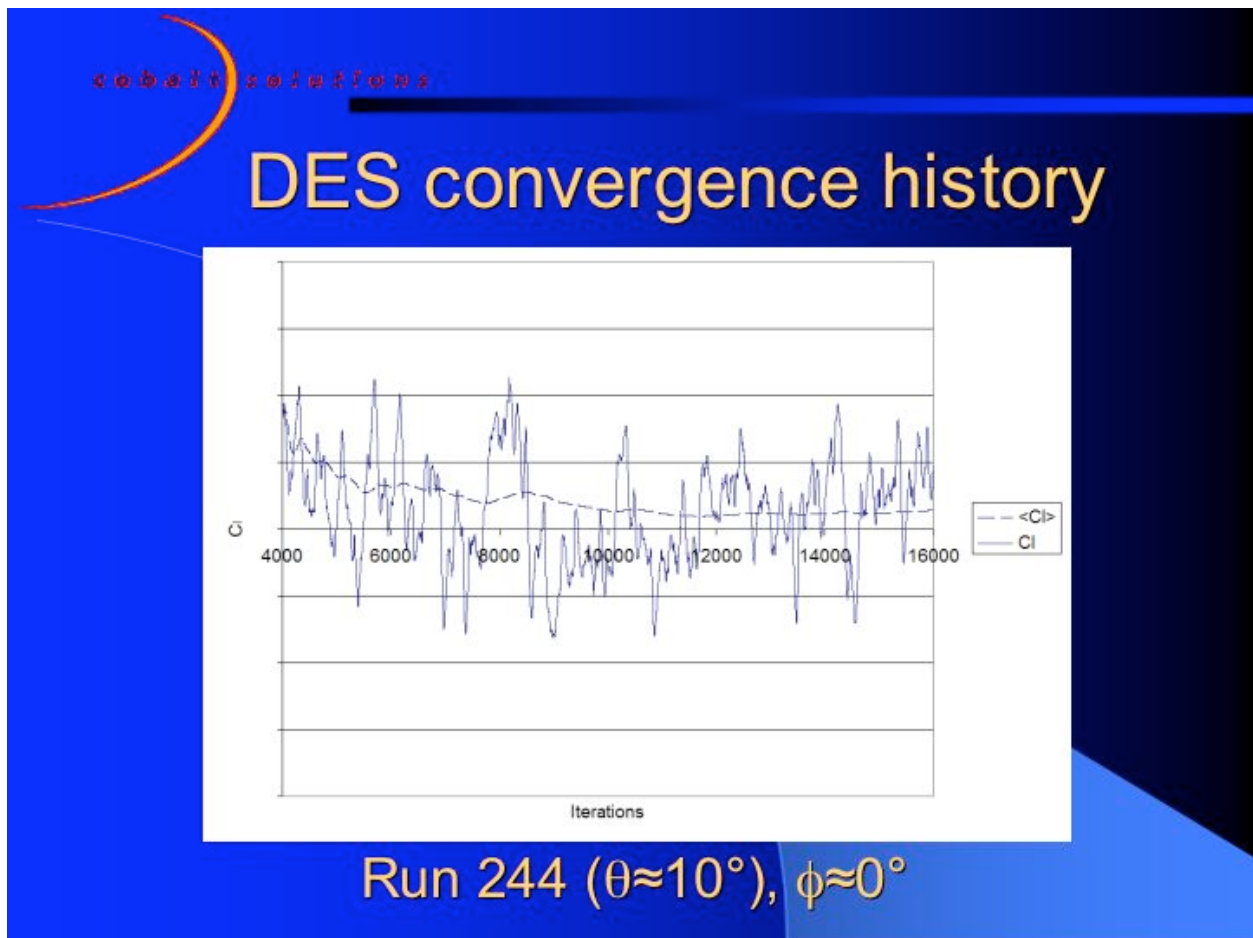
Run 242 ($\theta \approx 9^\circ$), $\phi \approx 30^\circ$
SST



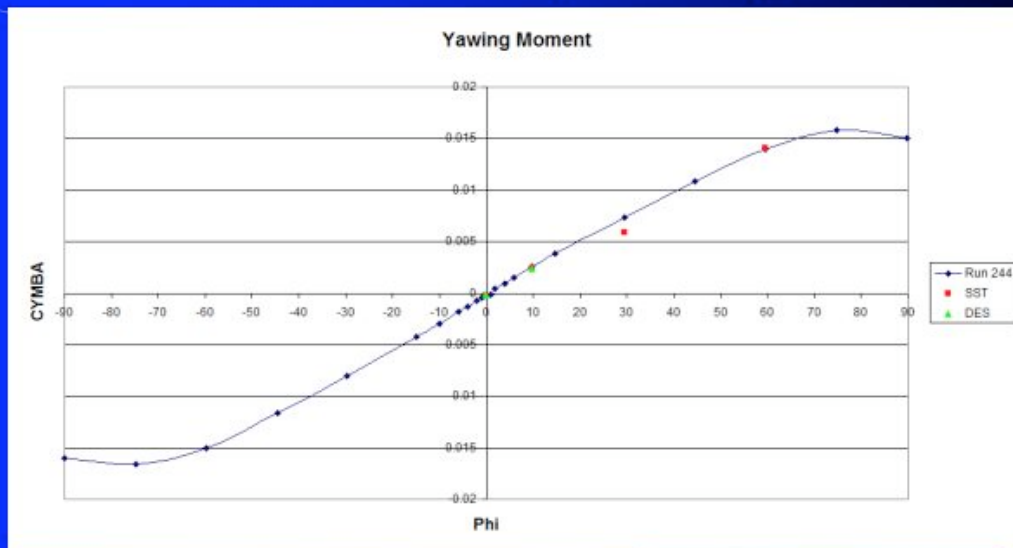
Rolling moment offset predicted by DES – is the sample size large enough?



Looks like enough samples have been taken to well define rolling moment. However more might change the time-averaged rolling moment some.



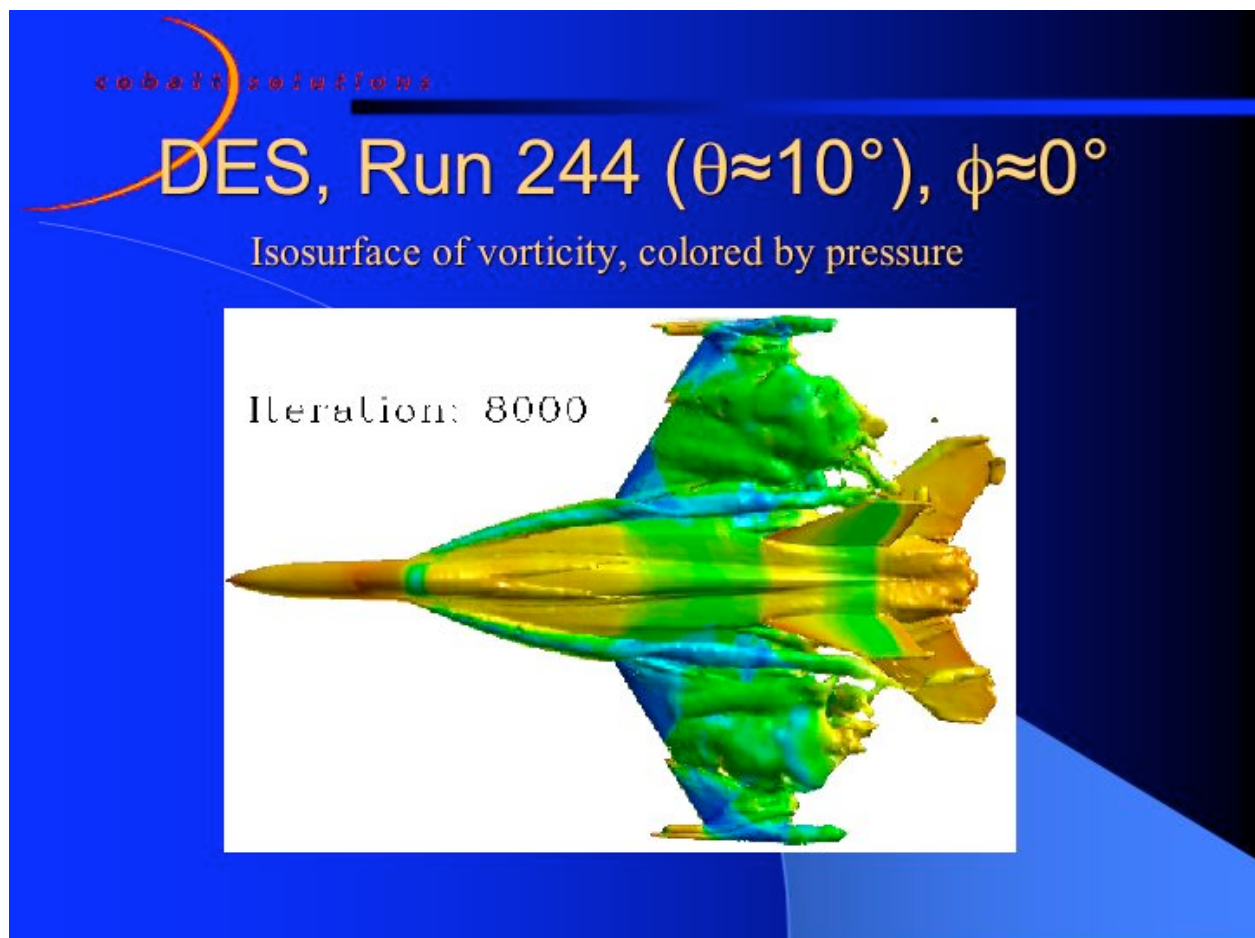
Run 244 ($\theta \approx 10^\circ$)



Run 244 ($\theta \approx 10^\circ$)



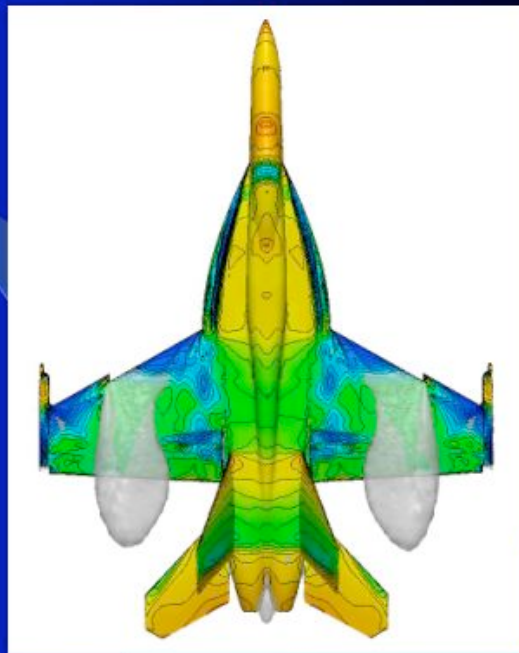
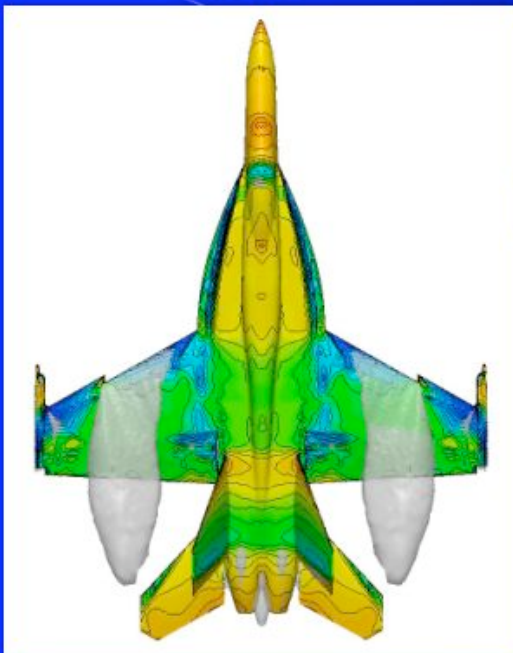
Unsteadiness now is due to separation moving from leading to trailing edge of the leading edge flap.



Run 244 ($\theta \approx 10^\circ$), $\phi \approx 0^\circ$

SST

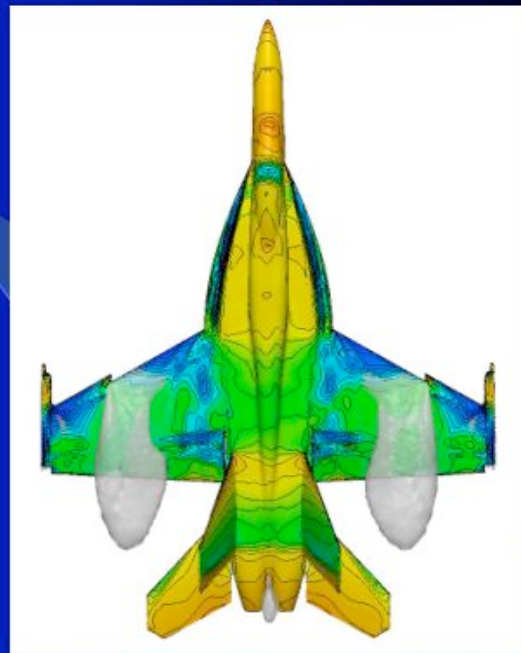
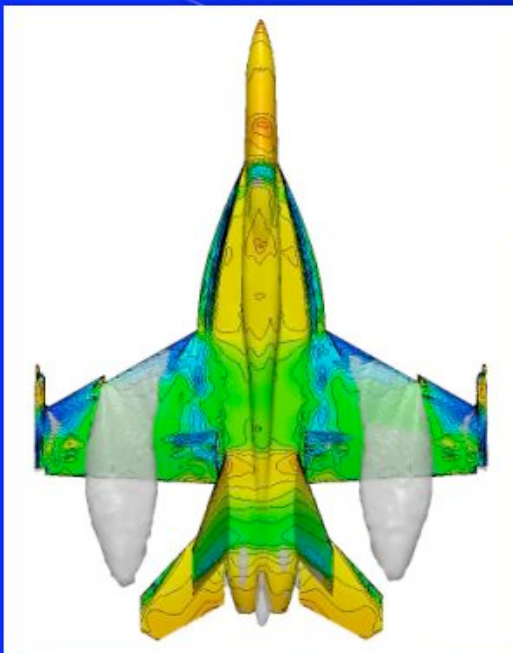
DES



Run 244 ($\theta \approx 10^\circ$), $\phi \approx 10^\circ$

SST

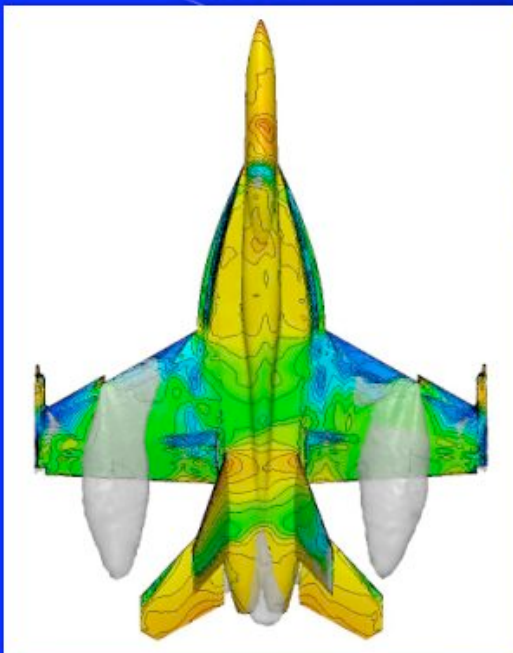
DES



ceba solutions

Isosurface of $u=0$, surface colored by pressure

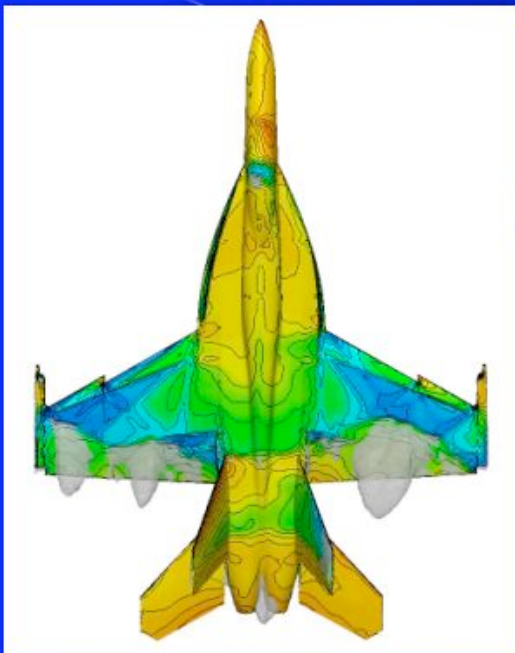
Run 244 ($\theta \approx 10^\circ$), $\phi \approx 30^\circ$
SST



ceba solutions

Isosurface of $u=0$, surface colored by pressure

Run 244 ($\theta \approx 10^\circ$), $\phi \approx 60^\circ$
SST



Oscillating Cases

ALE = Arbitrary Eulerian/Lagrangian.

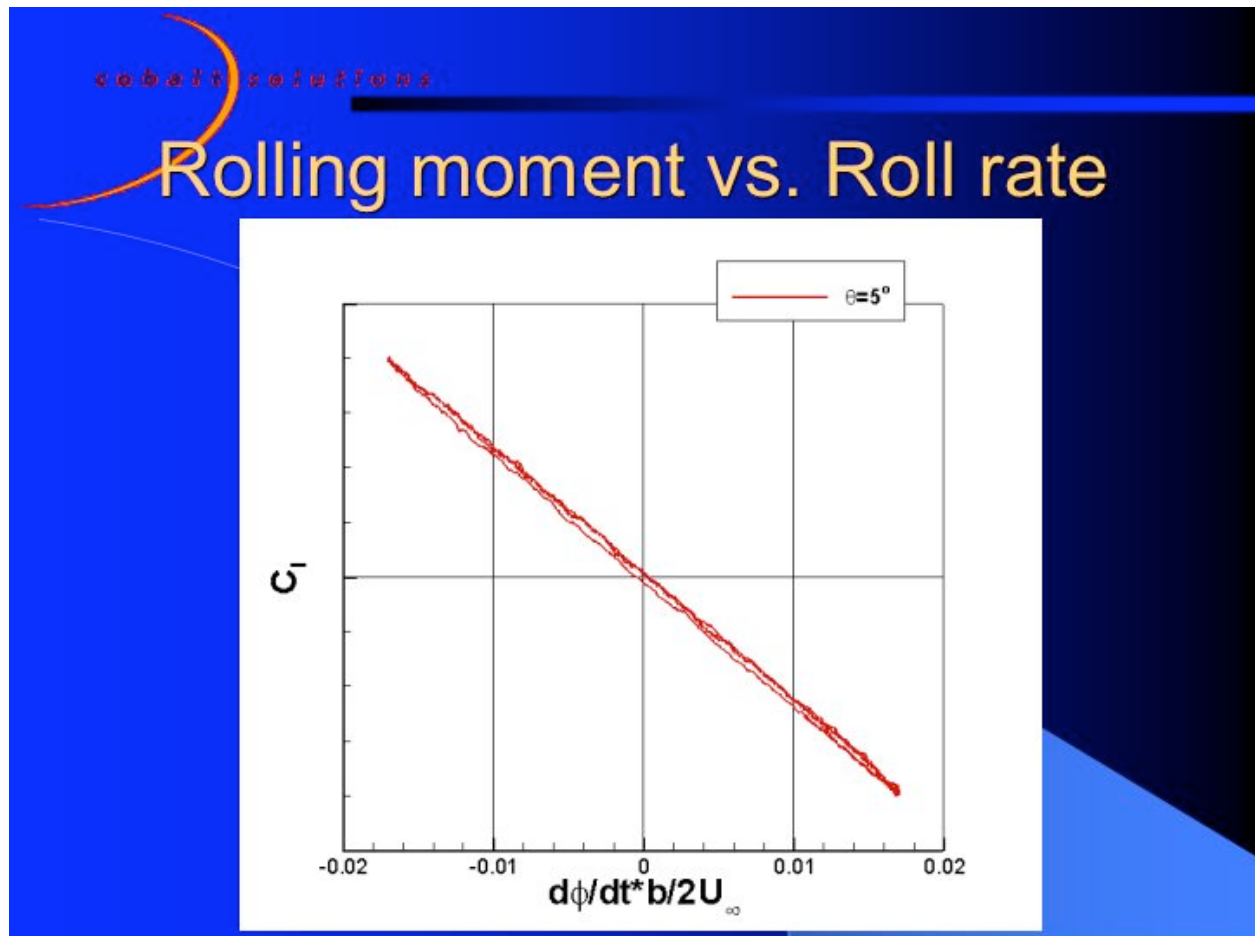


Solution Procedure

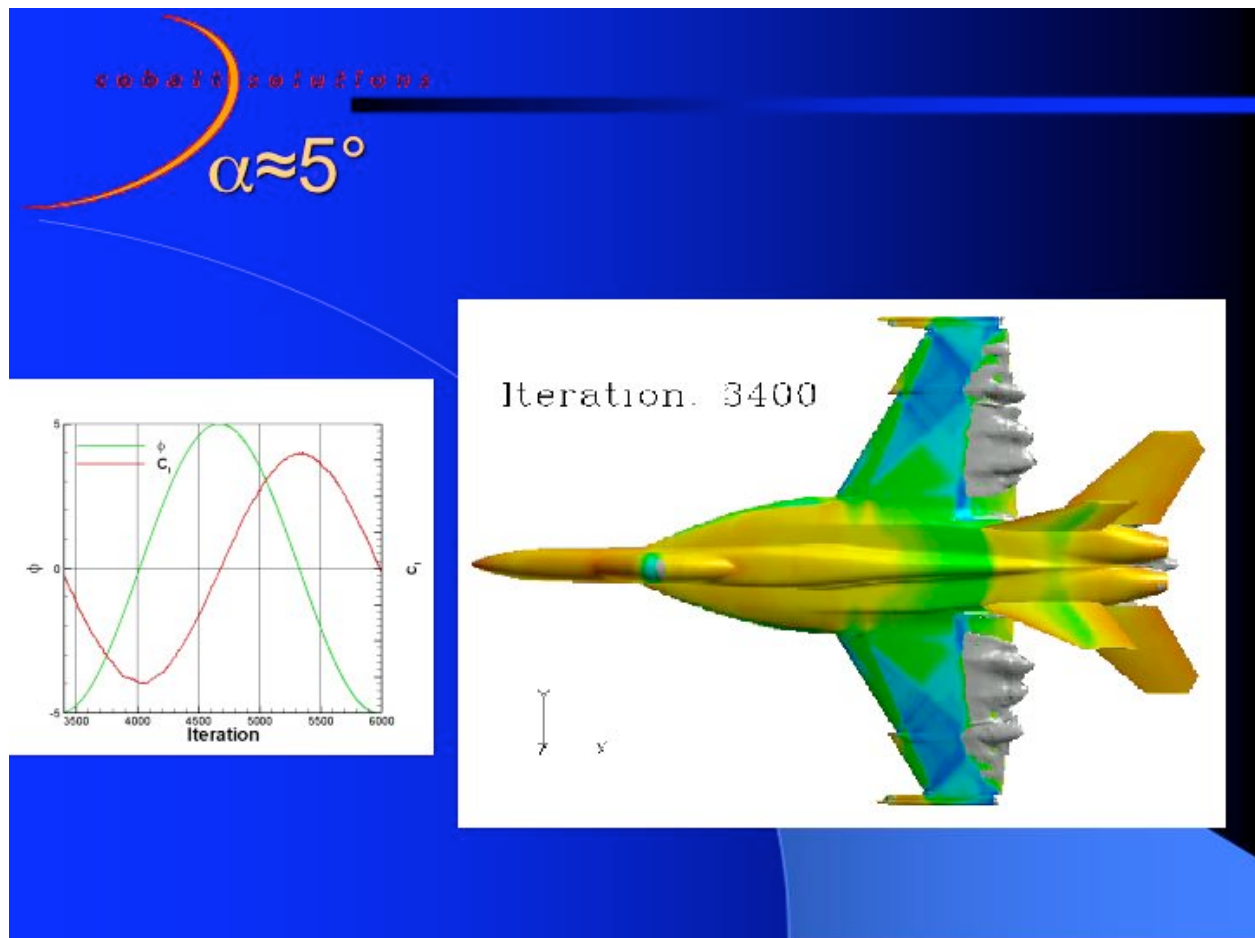
- Time-accurate with ALE formulation for grid motion
 - 5 Newton sub-iteration (for accurate grid motion)
 - $\Delta t^*=0.02$ (ran several timesteps to demonstrate timestep convergence)
- Prescribed sinusoidal oscillation around longitudinal axis
 - $\tan^{-1}(f^*)=1^\circ$, $f^*=0.0174$
 - 2,600 iterations per cycle
 - 4,000 cpu-hours per cycle
 - $\pm 5^\circ$ oscillation
- Menter's SST RANS model

$$\tan^{-1}\left(\frac{fb}{2U_\infty}\right) = \tan^{-1}(f^*) = 1^\circ$$

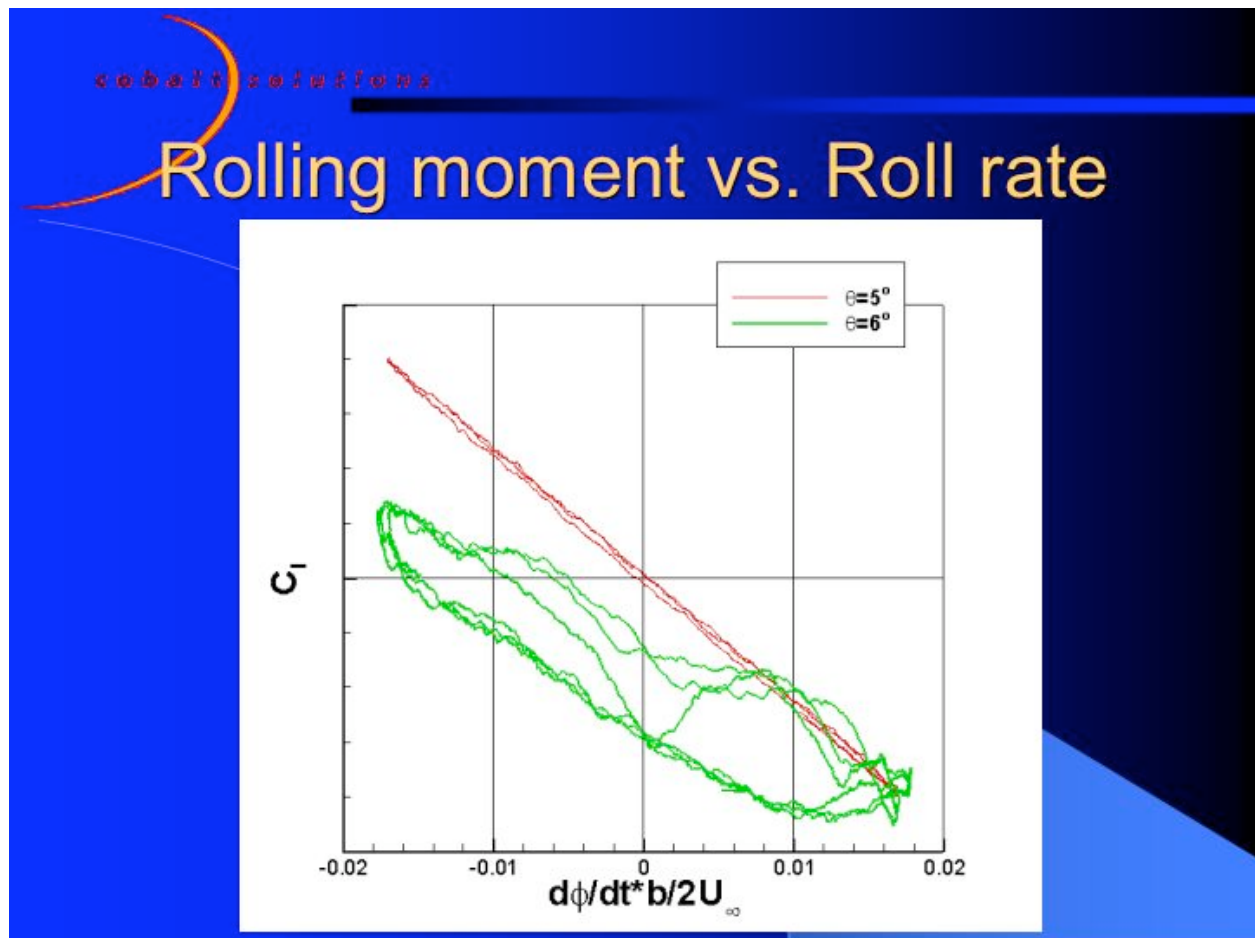
Linear and well behaved. Stable roll damping.



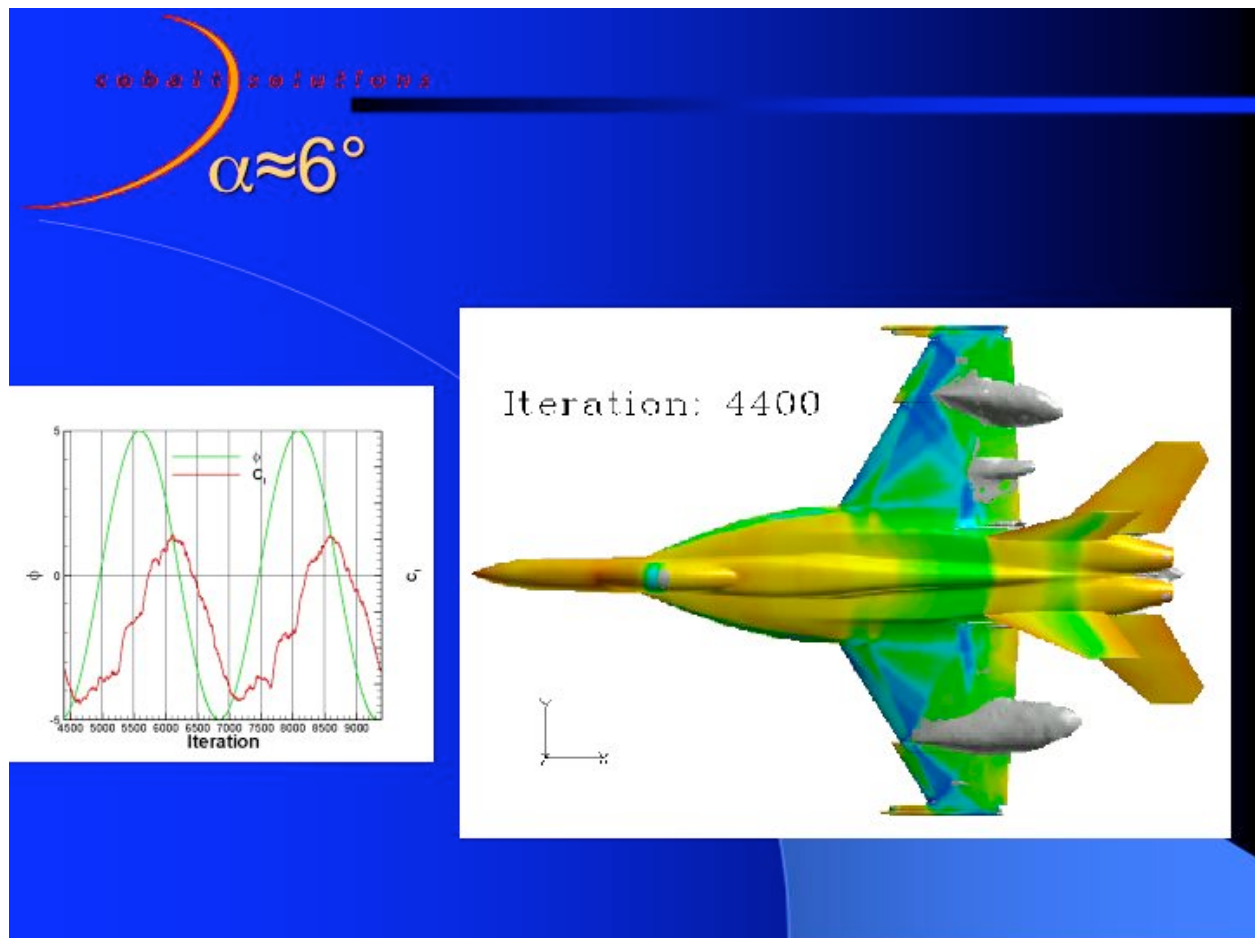
Separation at trailing edge – flow well behaved.



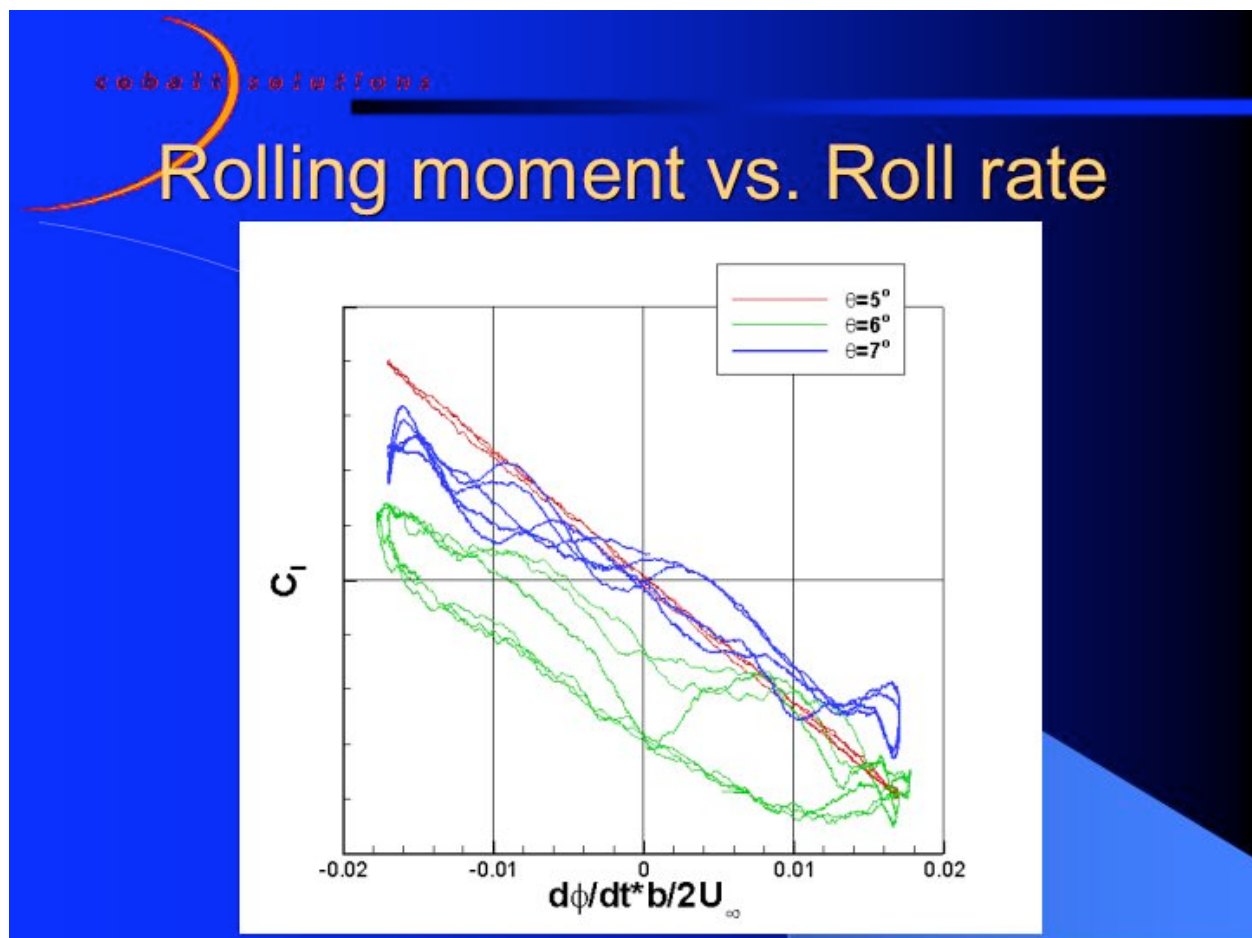
Large rolling moment offset. Several cycles run with varied timestep, but offset remained.



Offset due to differences in separation location. Hysteresis?

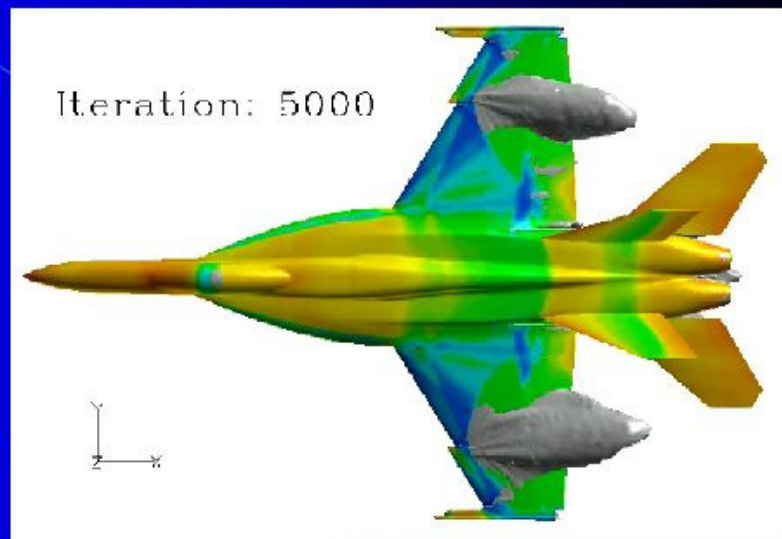
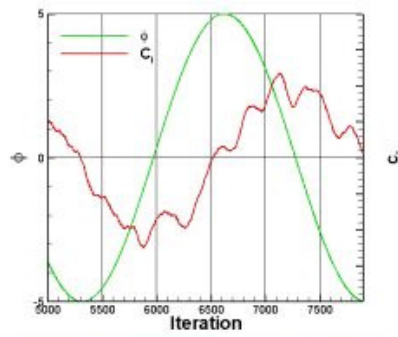


Slightly chaotic behavior, but linear and stable roll damping.

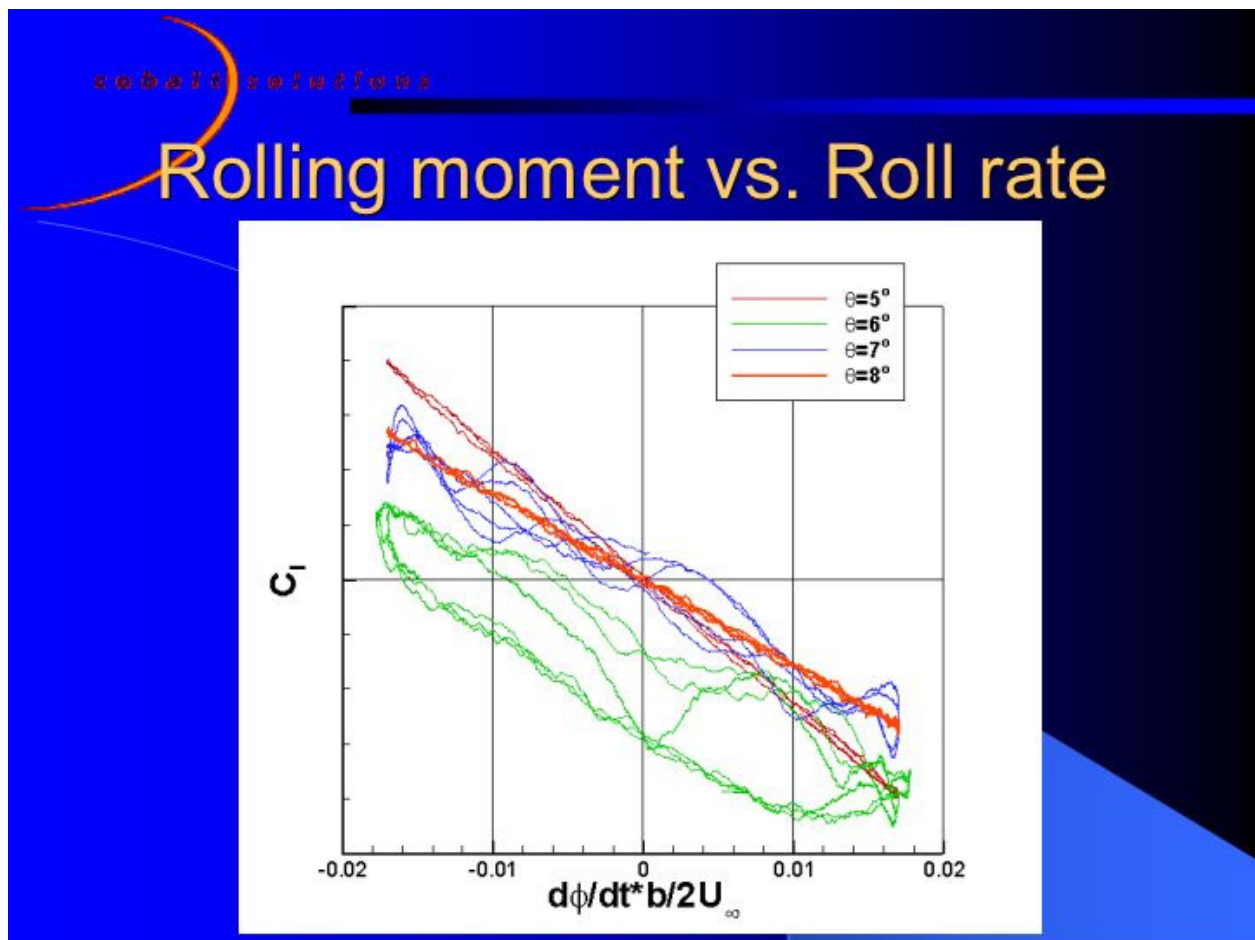


convergence

$$\alpha \approx 7^\circ$$

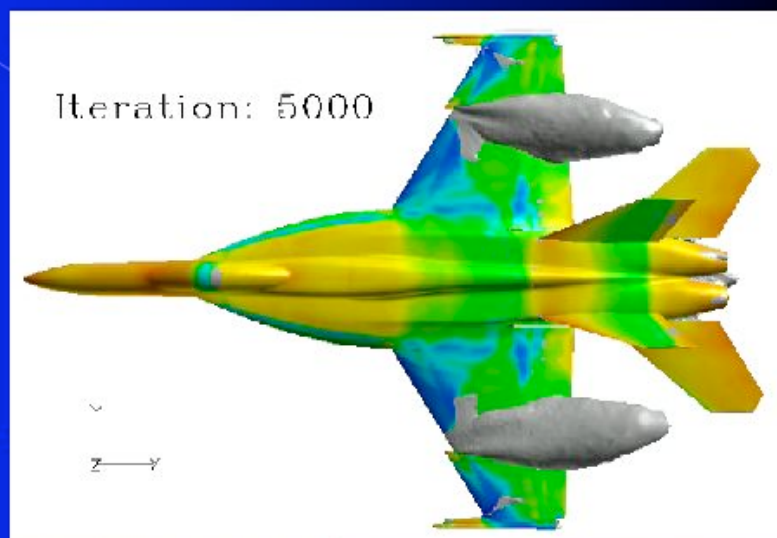
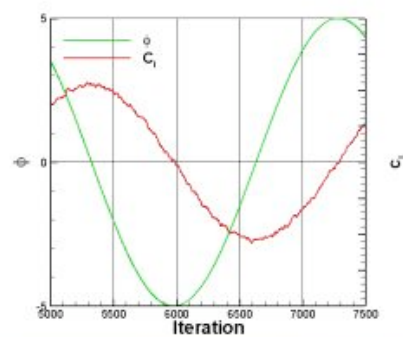


Positive roll damping. Note lowered slope – due to lower lift curve slope once shock moves forward on the wing.

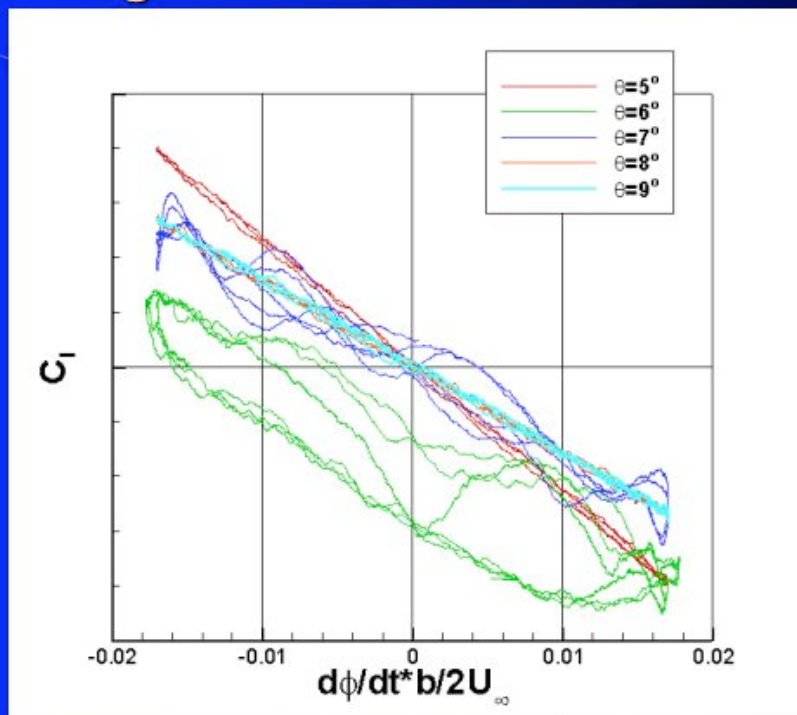


cobalt solutions

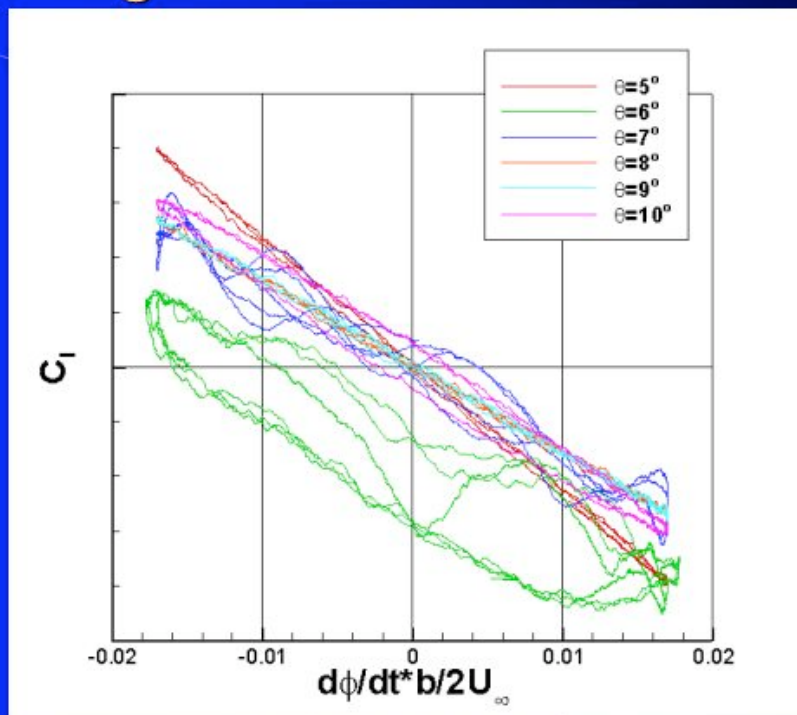
$$\alpha \approx 8^\circ$$



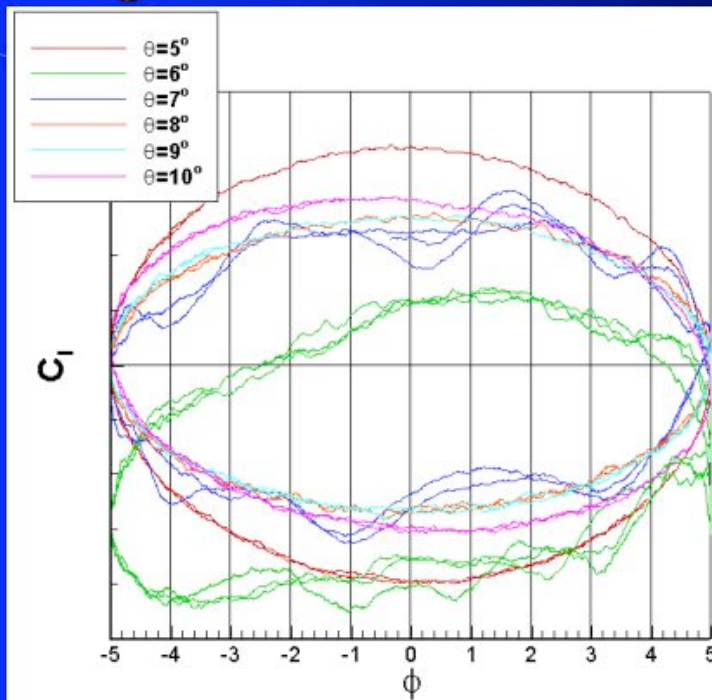
Rolling moment vs. Roll rate



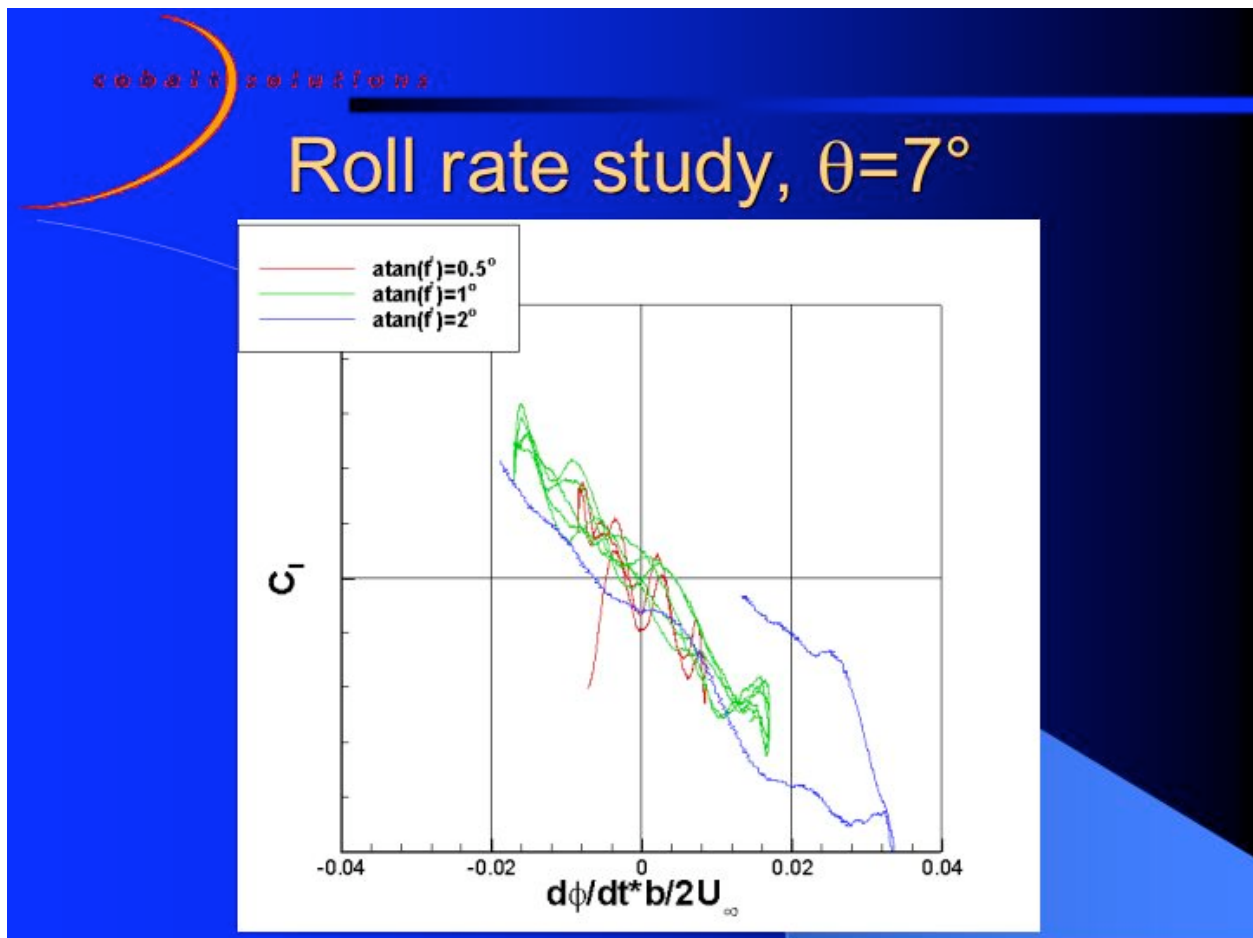
Rolling moment vs. Roll rate



Rolling Moment vs. Roll angle



Study still underway – not enough samples. Looking at dependence of roll damping on roll rate.



Conclusions

- RANS and DES applied to predict static stability derivatives in roll in AWS regime
 - DES showed better lift and moment predictions
 - Yawing moments and side force well predicted by both methods
 - Rolling moment more sensitive (both for CFD and wind tunnel)
- Prescribed rolls used to look at roll damping (RANS only)
 - All cases were stable in roll, but in AWS regime had more chaotic behavior. For one angle there was a significant rolling moment offset
 - Comparison to experiments still ongoing
- Continuing work
 - DES of prescribed rolls
 - More iterations on varying roll rate

This presentation discusses the requirements for and the ramifications of including unsteady aerodynamics and structural flexibility in the computation of stability and control derivatives for modern flight vehicles.

**APPLICATION OF COMPUTATIONAL
STABILITY AND CONTROL TECHNIQUES
INCLUDING UNSTEADY AERODYNAMICS
AND AEROELASTIC EFFECTS**

David M. Schuster
John W. Edwards
Aeroelasticity Branch
NASA Langley Research Center

NASA Symposium on Computational Methods for
Stability and Control (COMSAC)
September 23-25, 2003

The motivation behind the inclusion of unsteady aerodynamics and aeroelastic effects in the computation of stability and control (S&C) derivatives will be discussed as they pertain to aeroelastic and aeroservoelastic analysis. This topic will be addressed in the context of two applications, the first being the estimation of S&C derivatives for a cable-mounted aeroservoelastic wind tunnel model tested in the NASA Langley Research Center (LaRC) Transonic Dynamics Tunnel (TDT). The second application will be the prediction of the nonlinear aeroservoelastic phenomenon known as Residual Pitch Oscillation (RPO) on the B-2 Bomber. Techniques and strategies used in these applications to compute S&C derivatives and perform flight simulations will be reviewed, and computational results will be presented.

OUTLINE



- **Introduction**
 - Motivation for inclusion of unsteady aerodynamics and aeroelastic effects in computation of stability and control derivatives.
- **Applications**
 - Supersonic Transport (SST) aeroelastic wind tunnel model.
 - B-2 Residual Pitch Oscillation (RPO)
- **Conclusion**

9/24/2003 Schuster/Edwards

Within the LaRC Aeroelasticity Branch (AB), there are two primary objectives supporting the computation of stability and control derivatives. The first is to support free-flying cable-mounted aeroelastic and aeroservoelastic wind tunnel investigations in the TDT. The second is to support full-scale aeroelastic and aeroservoelastic analyses of modern flight vehicles. In the former case, since wind tunnel models are often conceptual in nature, a large database describing the model's flight characteristics, as is usually assembled for full-scale aircraft, is not available. Therefore, we rely virtually exclusively on empirical, analytical, and computational methods to predict the S&C performance of the model. This includes a requirement to predict both static and dynamic derivatives as well as the impact of structural flexibility on the model's performance. Since the TDT is a transonic facility, nonlinear aerodynamics is also an important

MOTIVATION AND REQUIREMENTS



- Support free-flying cable mounted aeroservoelastic wind tunnel investigations.
- Support full-scale vehicle aeroservoelastic analyses including automated flight control.
- Requirements include static and dynamic derivatives including flexibility, static and dynamic control derivatives, and simulations including nonlinear aerodynamics.



9/24/2003 Schuster/Edwards

contributor to the analysis. The widespread use of automated flight controls on virtually all modern commercial and military aircraft has introduced a new class of problems where the vehicle control system can interact with the aerodynamics and structural flexibility of the system. The discipline investigating these interactions is known as aeroservoelasticity and is rapidly growing in importance for prediction of on- and off-design vehicle performance. To effectively predict aeroservoelastic problems, accurate computation of control effectiveness is a must.

The first application to be discussed is the prediction of S&C derivatives for a SST wind tunnel model tested in the LaRC TDT. The model is an aeroelastically scaled model of a 1970's SST concept. It was developed to investigate control laws for the aircraft. The cable system employed in the TDT provides a five-degree-of-freedom mount for the model. A single vertical cable runs from the wind tunnel ceiling to the wind tunnel floor through a pair of vertically-mounted pulleys installed in the model just forward of the center of gravity. Similarly a single cable runs between the sidewalls of the wind tunnel through a horizontally mounted pair of pulleys aft of the center of gravity. In addition, four snubber cables run from the corners of the tunnel to the model near the C.G. These four cables can be interactively tightened and loosened. In the tight configuration, they are used to hold the model at the center of the tunnel during wind

SST APPLICATION



- Cable-mounted low-speed aeroelastic model tested in LaRC Transonic Dynamics Tunnel (TDT).
 - 5-DOF mount simulating pitch, plunge, roll, yaw, and side translation.
 - Hydraulically actuated inboard and outboard ailerons, flying horizontal stabilizer with a geared elevator.



9/24/2003 Schuster/Edwards

off conditions, and they are slack during “free-flight” testing. They can also be rapidly tightened when the vehicle encounters an instability to attempt to stabilize the aircraft. The model also includes hydraulically actuated wing and horizontal tail control surfaces. Control effectiveness derivatives, including flexibility effects are required to design the flutter-suppression control laws that are the subject of the test. In addition, the precise position and tension on the cables is defined using a computer program known as GRUMCBL, which requires S&C derivatives for the model.

This slide discusses the wind tunnel test objectives and requirements for S&C derivatives to support the test, emphasizing the importance of COMSAC techniques for this type of testing.

COMSAC IMPORTANCE



- Test objectives:
 - Investigate vehicle flutter characteristics with nearly rigid body degrees of freedom.
 - Long, thin design can lead to so-called body-freedom flutter.
 - Investigate flutter suppression active control laws.
- Need accurate S&C characteristics to assess stability of model during testing and to design flutter suppression control laws.
 - Static derivatives available through balance testing.
 - Dynamic derivatives available only from theoretical methods.

9/24/2003 Schuster/Edwards

This slide shows a video clip of the SST testing in which an aggressive flutter suppression control law is activated on the model. The model experiences a severe upset and the tunnel bypass valves and snubber support cables are activated. Unfortunately the model upset is too severe, and nonlinearity in the cable mount system and/or aerodynamics slowly drive the model to destruction. While it is unreasonable to blame the destruction of the model on poor predictions of S&C derivatives, this is a stark example of the importance of accurate predictions of these types of derivatives for this type of testing.

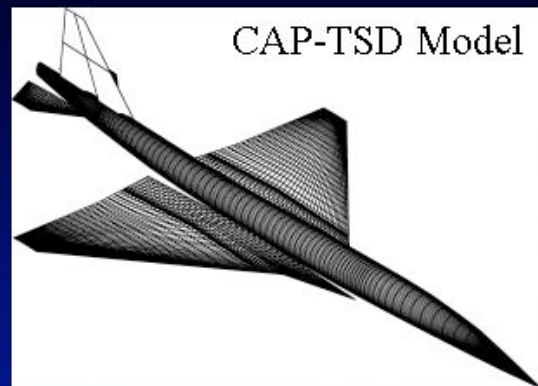


S&C derivative predictions for the SST model came from four primary sources, Stability and Control DATCOM, linear doublet lattice, transonic small disturbance potential flow (CAP-TSD), and wind tunnel balance data. Static, dynamic, rigid and flexible derivatives were developed for this configuration. Analyses using the Computational Aeroelasticity Program – Transonic Small Disturbance (CAP-TSD) are the focus of this presentation. Using this methodology, static derivatives were computed using a finite difference technique, but are not the main focus of this discussion. Dynamic derivatives were estimated by pulsing the configuration in pitch, plunge, yaw, and spanwise translation. Roll rate derivatives were computed using a steady analysis and imposing specialized boundary conditions to the lifting surfaces which represent the rolling motion of the aircraft.

ANALYSIS TECHNIQUES



- Static, dynamic, rigid, and flexible derivatives estimated using a number of methods.
 - DATCOM, linear doublet lattice, CAP-TSD, and wind tunnel balance data.
- CAP-TSD analysis focus of this presentation.
 - Static derivatives computed by finite difference,
 - Dynamic derivatives computed by pulse analysis.
 - Roll rate derivatives computed by a steady analysis with the rolling motion superimposed on the lifting surface boundary condition.



9/24/2003 Schuster/Edwards

This slide describes the essential features of the inviscid and viscous/inviscid interaction versions of CAP-TSD.

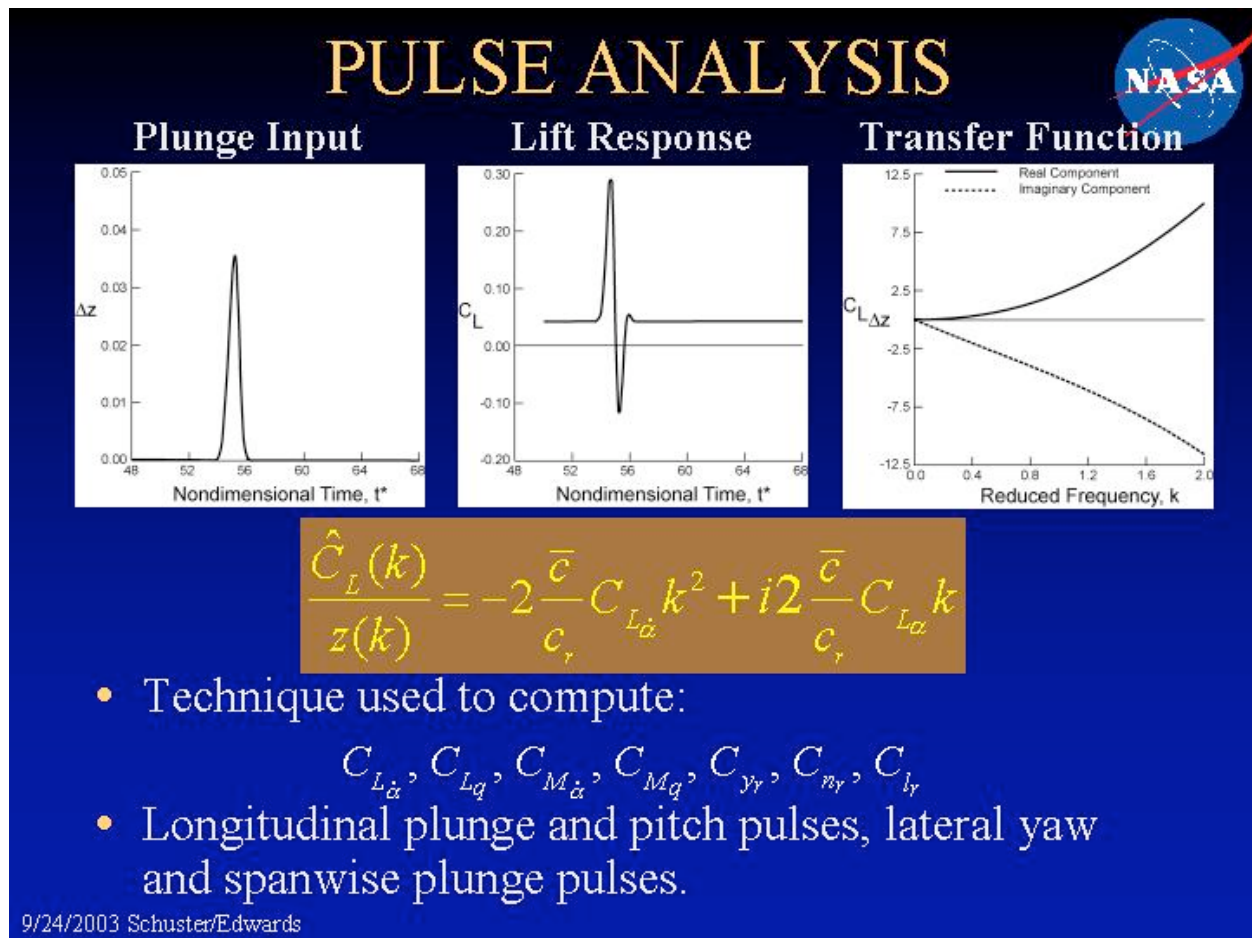
CAP-TSD



- Computational Aeroelasticity Program – Transonic Small Disturbance
 - Unsteady, transonic small disturbance potential flow.
 - Inviscid and viscous/inviscid interaction.
 - Interactive inverse integral boundary layer capable of analyzing separation onset and mildly separated flows.
 - Swept, tapered horizontal surfaces, rectangular vertical surfaces, fuselage, bodies, and horizontal and vertical control surfaces.
 - Configuration and component rigid body dynamics.
 - Structural flexibility for static and dynamic aeroelastic analysis.
 - Small disturbance assumptions allow structural and rigid body dynamics to be simulated without grid motion.

9/24/2003 Schuster/Edwards

Longitudinal and lateral rate derivatives were computed using a pulse analysis. This slide represents a configuration plunge pulse, the lift coefficient response to the pulse, and the transfer function computed from the input and response. The transfer function is derived by dividing the complex Fourier transform of the response by the transform of the input. The character of the transfer function at zero frequency defines the static lift curve slope and the dynamic S&C derivative due to angle-of-attack rate. A pitch pulse of the configuration results in a combined pitch rate and angle-of-attack rate derivative, which in conjunction with the plunge pulse can be used to extract the pitch rate derivative. A similar procedure is used to compute the lateral derivatives due to yaw rate and sideslip rate.



This slide shows the longitudinal and lateral rate derivatives computed by CAP-TSD and compared with results from doublet lattice and DATCOM. While the various methods show a general agreement in magnitude and sign between the methods, one is hard-pressed to say the correlation for this case is good. In general, CAP-TSD tends to over predict the magnitude of the rate derivatives, with the exception of pitching moment. There are several modeling assumptions inherent in each of the methods which could have a profound impact on the results, but given the time constraints and objectives of the analysis, it was impossible to investigate these issues. Certainly, further investigation of techniques for computing these derivatives is warranted before widespread acceptance of the methodology can be anticipated.

SST Dynamic Derivatives

$M = 0.50$

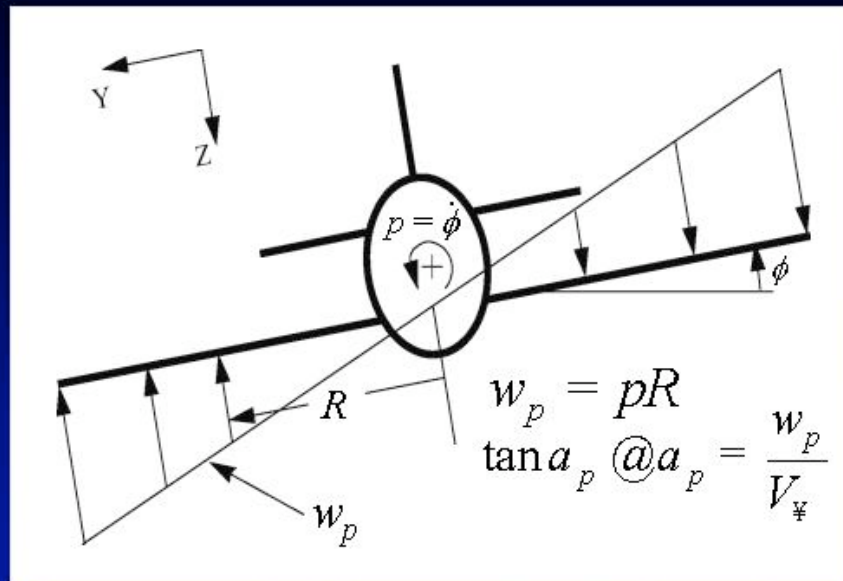


Derivative	Method		
	CAP-TSD	Doublet Lattice	DATCOM
$C_{L\dot{\alpha}}$	1.080	n/a	0.606
C_{Lq}	4.596	n/a	3.655
$C_{M\dot{\alpha}}$	-0.610	n/a	-0.962
C_{Mq}	-1.193	n/a	-2.354
C_{Yr}	0.952	0.467	n/a
C_{nr}	-0.688	-0.290	-0.288
C_{lr}	0.135	0.063	0.056

9/24/2003 Schuster/Edwards

Roll rate derivatives were computed using CAP-TSD by modifying the lifting surface boundary conditions used in the code to represent a steady rolling motion of the vehicle. Incorporation of the rolling motion in this manner allows roll rate derivatives to be computed using a steady analysis as opposed to a time-accurate computation.

Roll Rate Derivative Analysis



- Steady downwash/sidewash distribution added to horizontal and vertical surface boundary conditions.
- Requires only a steady computation.

9/24/2003 Schuster/Edwards

The roll rate derivatives computed by CAP-TSD using this technique are in much better agreement with doublet lattice and DATCOM than were the previous longitudinal and lateral rate derivatives. The exception being yawing moment due to roll rate, which is a historically-difficult derivative to estimate.

Roll Rate Derivatives

$M = 0.50$



Derivative	Method		
	CAP-TSD	Doublet Lattice	DATCOM
C_{l_p}	-0.206	-0.232	-0.207
C_{y_p}	-0.032	-0.022	0.014
C_{n_p}	0.025	0.014	-0.002

9/24/2003 Schuster/Edwards

Structural flexibility effects were also investigated for the aircraft by adding structural modes to the CAP-TSD model and performing an aeroelastic analysis of the SST configuration. This slide shows the first six structural modes and frequencies included in the aeroelastic analysis.

Elastic Effects



SST Mode Shapes on CAP-TSD Grid

Mode 1 - 2.70 Hz.



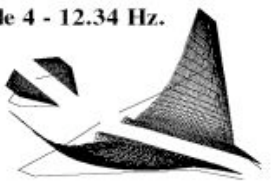
Mode 2 - 4.33 Hz.



Mode 3 - 7.70 Hz.



Mode 4 - 12.34 Hz.



Mode 5 - 15.35 Hz.



Mode 6 - 18.61 Hz.



- Elastic modes added to simulate effect of dynamic pressure on stability derivatives.
 - Elevator reversal predicted by both CAP-TSD and experiment.

9/24/2003 Schuster/Edwards

Lift curve slope, elevator effectiveness and outboard aileron effectiveness as computed by CAP-TSD are compared with balance data on the model acquired in the TDT prior to cable-mount testing. Due to aeroelastic deformations, these derivatives are a function of the dynamic pressure. Since the wind tunnel model is inherently flexible, no rigid data for the model on the balance is available. In general, the magnitude and trends in the data as compared to experiment are very good with the exception of the lift curve slope. CAP-TSD does not compute the wing and horizontal tail carry-over lift across the fuselage making the CAP-TSD lift curve slope lower than that of the experiment. An important feature to note is the loss in elevator and aileron control effectiveness with increasing dynamic pressure predicted by the theory and supported by the experimental data. Both the theory and experiment indicate an elevator reversal for this aircraft at a dynamic pressure between 20 and 30 psf.

Effect of Flexibility on Stability Derivatives, $M = 0.50$



			Dynamic Pressure (psf)		
Derivative	Data Source	Rigid	20	30	40
$C_{L_{\alpha}}$	CAP-TSD	2.590	2.060	1.930	1.810
	Experiment	n/a	2.430	2.270	2.300
$C_{L_{\delta_H}}$	CAP-TSD	0.330	0.079	-0.009	-0.080
	Experiment	n/a	0.077	-0.006	-0.130
$C_{l_{\delta_A}}$	CAP-TSD	0.046	0.024	0.019	0.016
	Experiment	n/a	0.030	0.028	0.025

- CAP-TSD calculates no fuselage or wing carry-over lift.
 - Accounts for discrepancy in lift curve slope due to angle of attack.

9/24/2003 Schuster/Edwards

In summary, this analysis represents a pure application of the available methodology with minimal opportunity to effectively research the methods employed or the results obtained. All of the data were used to establish bounds for the input data to the GRUMCBL cable-mount stability program to determine cable positioning and tensions for the free-flying test. There is considerable scatter in the derivatives produced by the various methods, particularly for the longitudinal and lateral dynamic derivatives. Structural flexibility was a significant player in this analysis, and both CAP-TSD and the experimental data indicated an elevator reversal at a relatively low dynamic pressure.

SST Analysis Summary



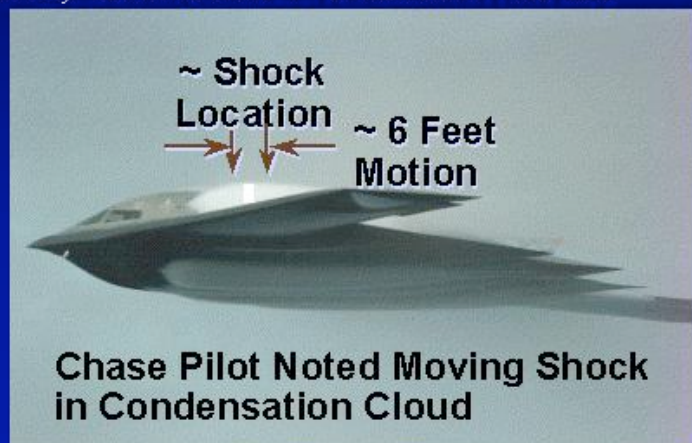
- Pure application of available methodology: minimal research into methods.
 - All data input to a cable stability analysis program (GRUMCBL) to define cable positions and tensions and determine model stability on the cable mount system.
- Considerable scatter in the data among the methods, particularly for estimation of dynamic derivatives.
- Structural flexibility was a significant player in this analysis.
- Cannot neglect ability to predict lateral derivatives.
 - Full-span analytical models.
- Availability of good benchmark data for future method development is an issue.

9/24/2003 Schuster/Edwards

B-2 Residual Pitch Oscillation

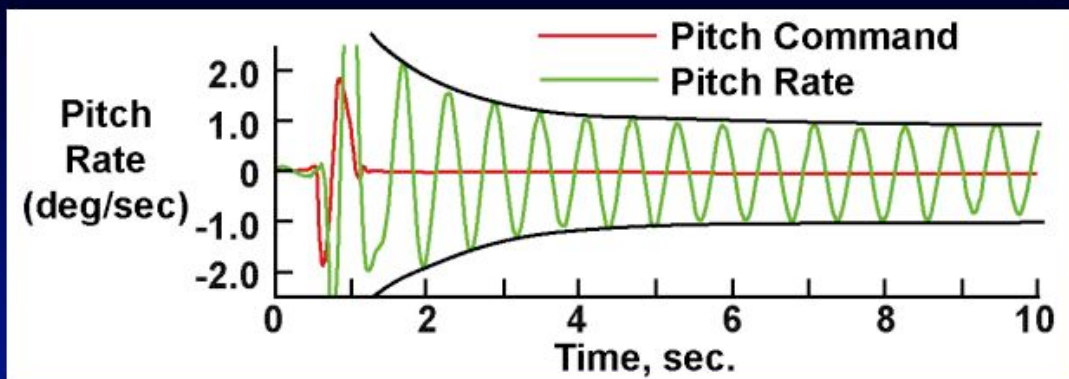


- CAP-TSDV modified to investigate problem.
 - Rigid body short period dynamics and trim added.
 - Flight control simulation added.
- Accurately modeled the heavyweight RPO event, but the lightweight event was not as accurately predicted.
- Reported in AIAA-98-1805, "Residual Pitch Oscillation (RPO) Flight Test and Analysis on the B-2 Bomber" and NASA/CP-199-209136 "Simulation of Non-Linear Transonic Aeroelastic Behavior on the B-2"



9/24/2003 Schuster/Edwards

Typical RPO Event



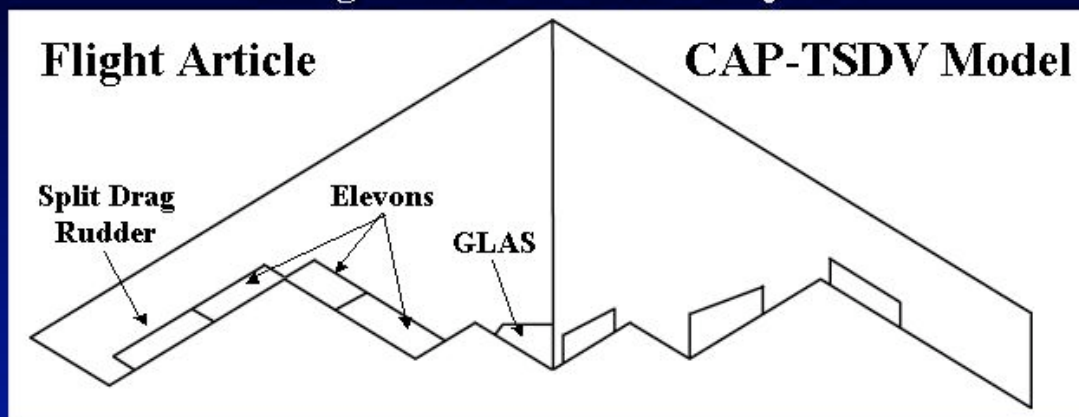
- Pitch command doublet results in an oscillatory pitch response that decays over time to a Limit Cycle Oscillation (LCO).
- Event involves shock motion and separated flow at transonic flight conditions, rigid body DOF, structural flexibility and nonlinear flight control system (servo-valve hysteresis).

9/24/2003 Schuster/Edwards

CAP-TSDV B-2 RPO Analysis



Longitudinal Control Layout



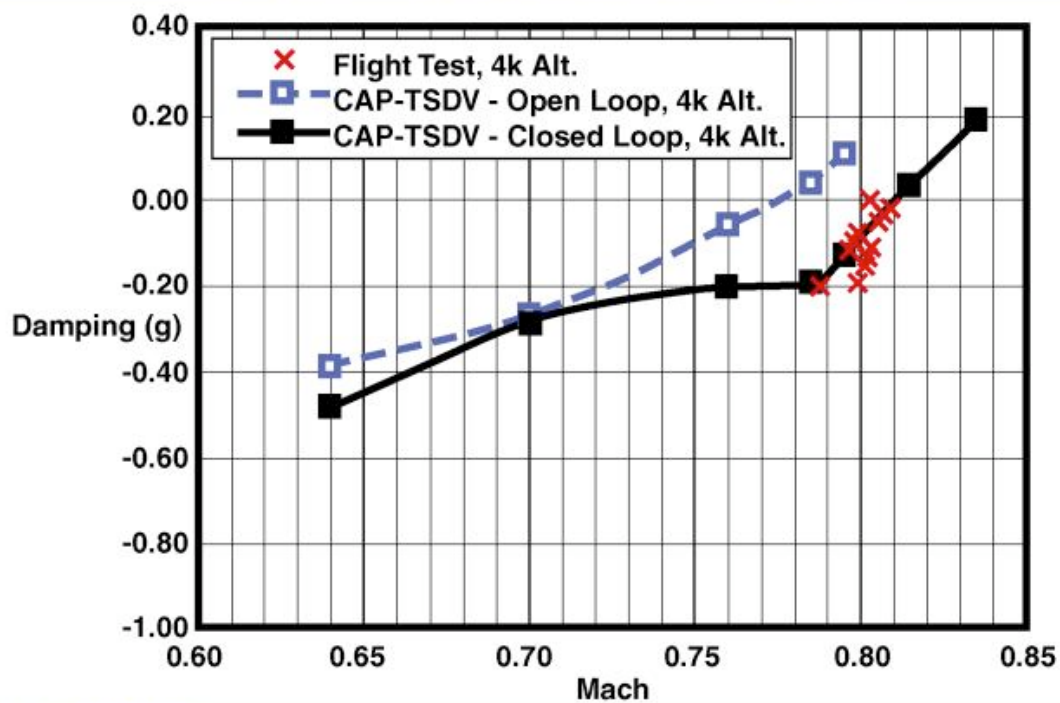
- Gust Load Alleviation System (GLAS), inboard elevon and outboard elevon primary control surfaces attempting to control the RPO during flight.
- CAP-TSD modeling restrictions impact the modeling of these control surfaces as well as the wing tip.

9/24/2003 Schuster/Edwards

Heavy Weight RPO Events



CAP-TSDV Comparison with Flight

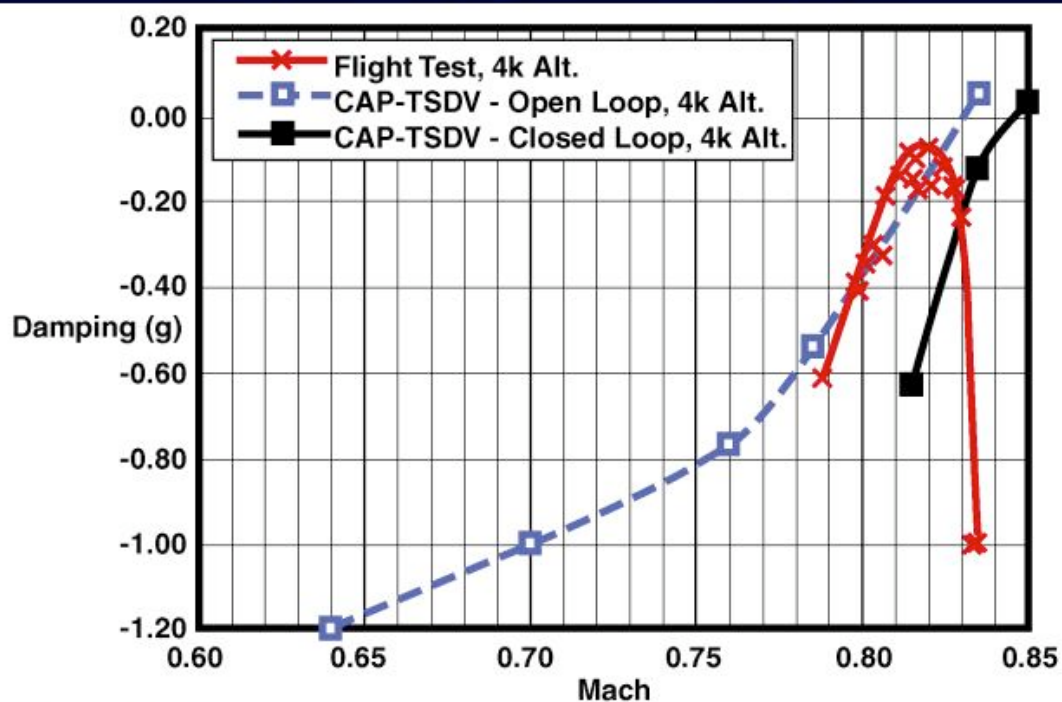


9/24/2003 Schuster/Edwards

Light Weight RPO Event



CAP-TSDV Comparison with Flight



9/24/2003 Schuster/Edwards

B-2 RPO Summary



- Probably the most comprehensive analysis of a modern flight vehicle ever attempted.
 - Transonic, separated flows.
 - Rigid body motion, structural flexibility and vehicle mass effects included in the analysis.
 - Nonlinear flight control system integrated with hysteresis.
 - CAP-TSDV in complete longitudinal control of the vehicle during the analysis.
- Prediction of the heavy weight RPO is very encouraging, but improved transonic/separated flow analysis required to accurately predict the light weight RPO events.

9/24/2003 Schuster/Edwards

Conclusion



- Examples demonstrate the types of analyses required to effectively analyze modern, flexible free-flying vehicles.
- Must be able to analyze in the corners of the flight envelope.
 - Transonic, separated, unsteady flows.
 - Structural flexibility cannot be neglected.
 - Many problems rooted in lateral control of the vehicle, requiring full-span simulations.
- For complex aeroservoelastic interactions, such as the B-2 RPO, estimation of S&C derivatives may not be good enough.
 - Nonlinear, CFD-in-the-loop simulation may be required.
- Correlation data for method development is an issue.

9/24/2003 Schuster/Edwards

This presentation is based upon AIAA Paper 2000-4325. The companion paper documenting the experimental portion of the study is AIAA Paper 2000-4016.



Hinge Moment Predictions Using CFD

M. Grismer, D. Kinsey, D. Grismer
Air Vehicles Directorate
Air Force Research Laboratory



Why?

- **Accurate predictions of aircraft control surface hinge moments needed for design**
 - Conservative predictions lead to heavier, more expensive actuators
 - Negatively impacts aircraft performance and handling
- **Accurate predictions are difficult to obtain due to nonlinear effects**
 - Boundary layer separation
 - Shock/boundary layer interaction
- **CFD has not been providing accurate predictions**
 - Recent study had prediction errors ranging from -40 to 55%
 - Discussions with industry engineers confirm errors of this magnitude common



Hinge Moment Study

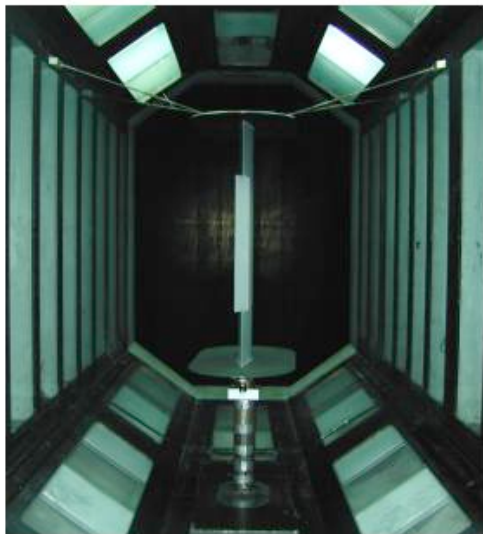
- **Air Vehicles directorate initiated a two part study to determine the factors leading to the large prediction errors**
- **Experimental investigation**
 - Provide an accurate database of hinge moment data
 - Used constant cross section, rectangular wing with partial span flap
 - Wing/flap section taken from discarded AF drone to reduce cost
- **Numerical investigation**
 - Compare against experimental database
 - Consider varying levels of modeling physics
 - Consider varying levels of geometric complexity

The model was held very securely by the mount and cables, so aeroelastic effects would be negligible. The wing was about 4 feet long with a 2 foot chord, and the tunnel cross section is 7 by 10 feet.



Experimental Setup

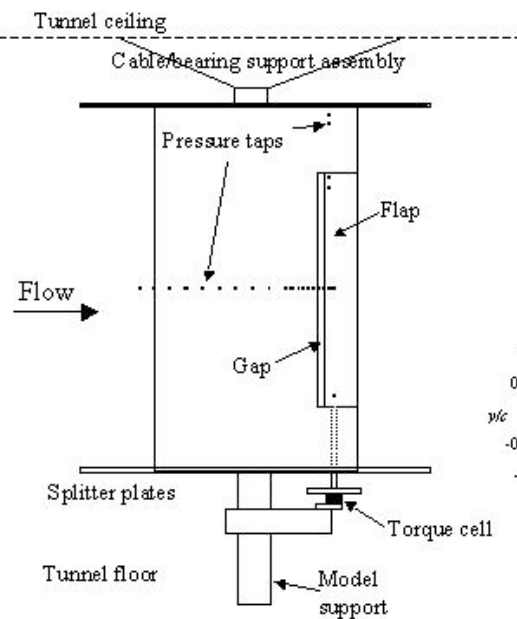
- Model setup in Subsonic Aerodynamic Research Laboratory (SARL) at WPAFB



The line of pressure taps in the figure above got displaced somewhat, they should begin at the leading edge of the wing and end about the trailing edge of the flap.

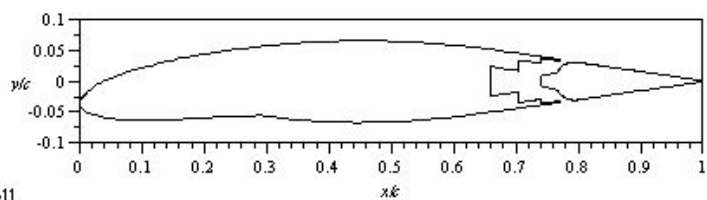


Experimental Measurements



Quantity	Range
M_∞	0.2, 0.3, 0.4, 0.5
$Re (\times 10^6)$	2.7–6.0
q_∞ (psf)	56.7, 122, 207, 304
α ($^\circ$)	-4, 0, 4, 8, 12 (some for 9)
Flap Gap Spacing (in.)	0.15, 0.25, 0.50
δ ($^\circ$)	-10, 0, 10, 15, 20

Experimental test matrix



Measured cross-section at $\alpha = 0^\circ$, $\delta = 0^\circ$

This work was done with the government version of the code, at the time there only was one version...



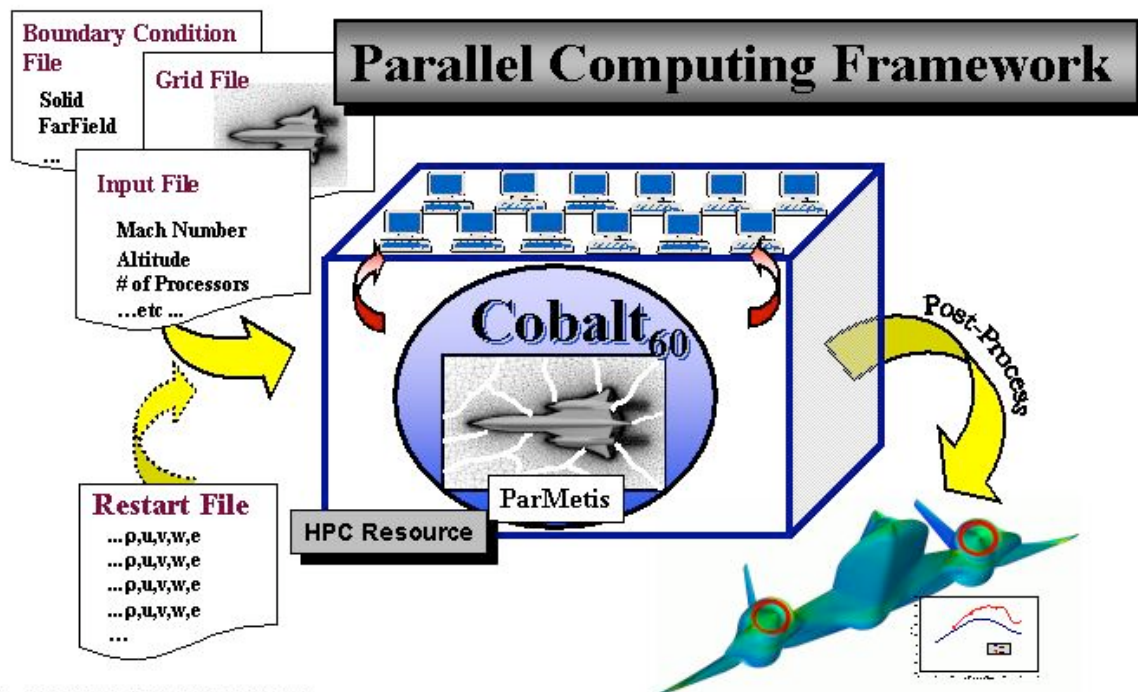
Numerical Study

- **Using Cobalt₆₀**
 - Unstructured Euler/Navier-Stokes solver
 - Cell-centered, finite-volume, Godunov-type solver
 - Second-order spatial accuracy using least-squares gradient reconstruction and limiting
 - Second-order point implicit time integration
 - Parallel (via MPI)
 - Arbitrary cell types
 - Turbulence models
 - » One equation Spalart-Allmaras (S-A)
 - » Two-equation Menter's shear-stress transport (SST)
 - » Others (Menter's baseline, DES)
- **Unstructured methodology allows for investigation of arbitrarily complex geometry in short amount of time**

- Parallel computing framework
- Required files: input,bc,grid
- Submit to machine: choosing the number of processors we want/require
- Automatically decomposes grid: ParMetis
- ParMetis developed at the Univ. of Minnesota funded by the Army
- Recombines into 1 zone when finished
- Can restart on a different number of processors (just change 1 input)



Parallel Computing Process



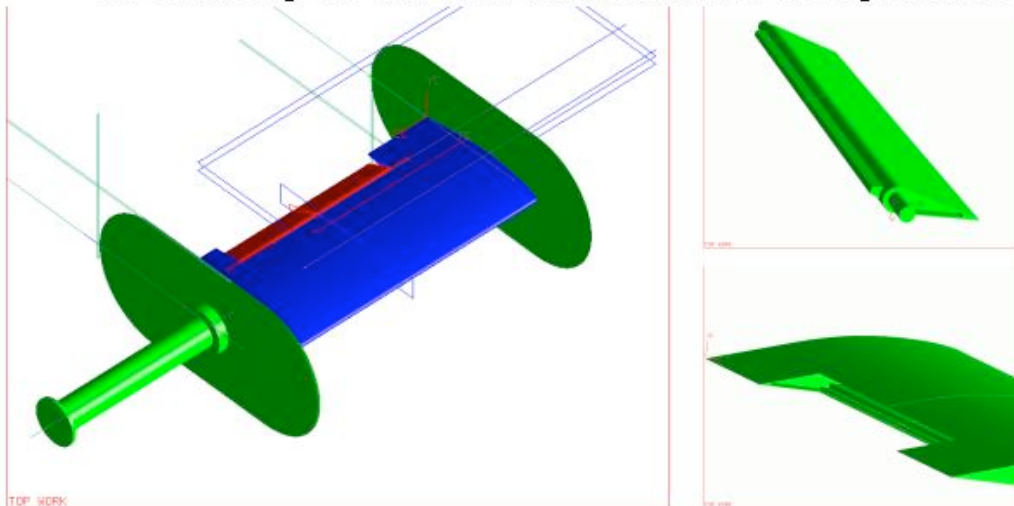
Slide courtesy of Ken Wurtzler

The difficulty in creating the digital model was defining the airfoil cross section in a way that was sufficiently smooth. Simply digitizing points of the wing was not sufficient as this lead to slope discontinuities between points. Tracing the model and fitting the trace with bezier cubic splines resulted in a continuous smooth digital representation. The main remaining difference between the digital model and the physical model is the fact that the actual wing is hollow, while the digital representation is solid. There are a number of holes and gaps, particularly in the flap area, that allow air to circulate inside the real wing.



Geometry

- **CAD geometry based upon careful measurement of experimental model**
 - **Bézier cubic splines used to make mathematical airfoil representation**

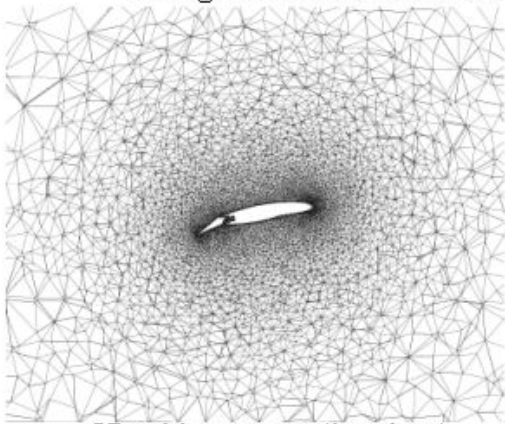


For the cases that included the tunnel walls and mount, viscous layers were only included around the wing, flap and end plates.

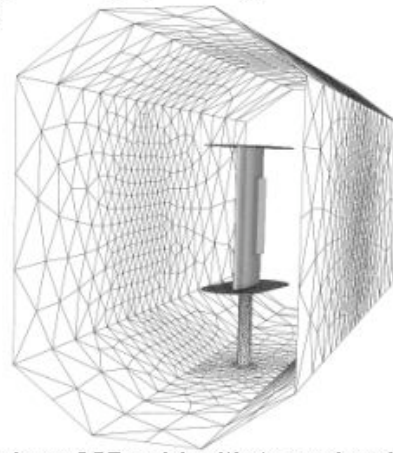


Grid Generation

- **2D unstructured grids generated with Tri2d**
- **3D unstructured grids generated with VGRIDns**
 - **Boundary layer tetrahedra combined into prisms with Blacksmith**
 - **Separate grids required for flap angle changes, and angle-of-attack changes when tunnel walls included**



3D grid cross-sectional cut



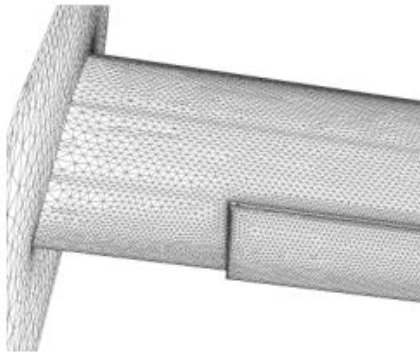
Boundaries of 3D grid with tunnel walls, mount



Grid Refinement

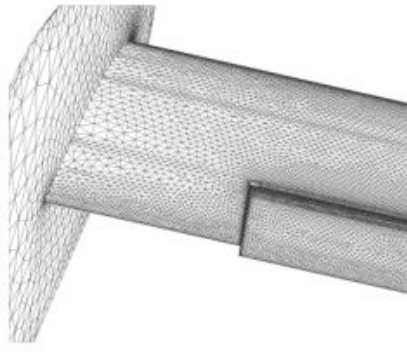
- Three levels of flap grid density used

Standard



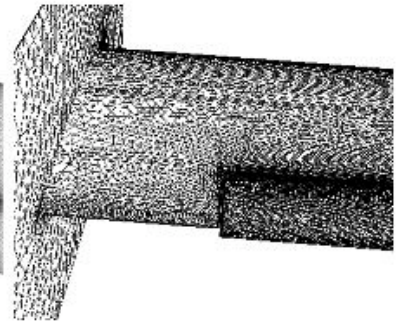
2.4 million cells

Medium



3.1 million cells

Fine



3.4 million cells



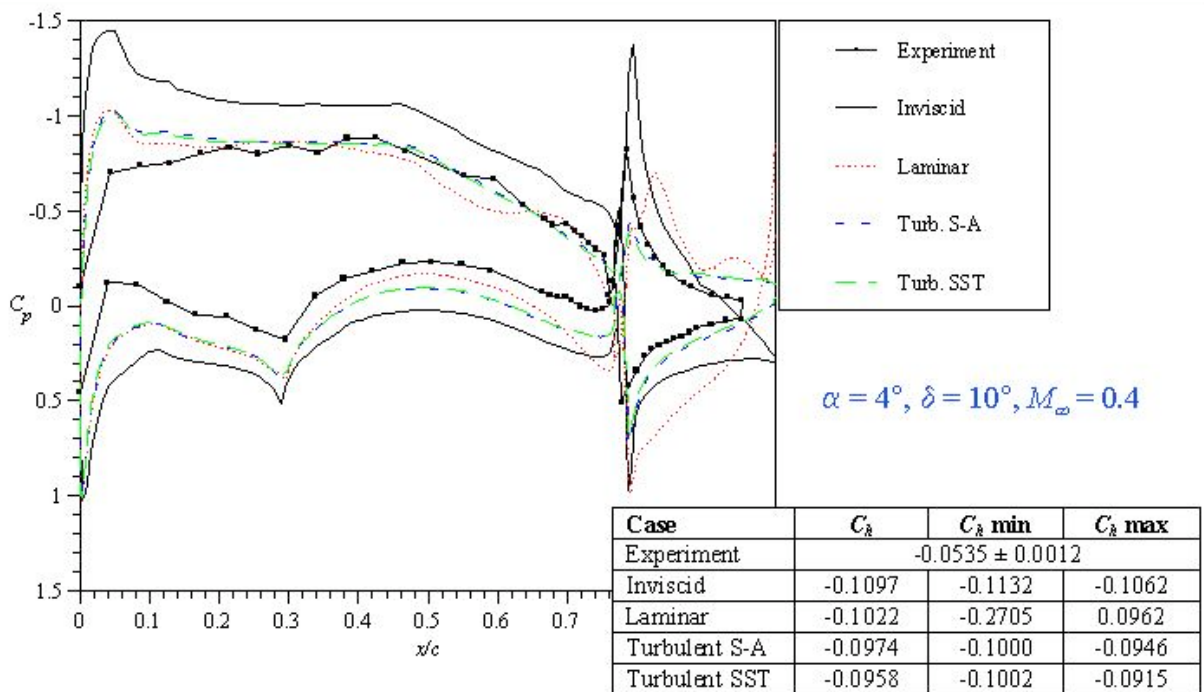
Results

- Considered two of the experimental test conditions
 - *Case 1:* $\alpha = 4^\circ$, $\delta = 10^\circ$, $M_\infty = 0.4$, flap gap = 0.15 inches
 - » “Get feet wet” - expected to match data for benign case
 - » Flow separation not indicated in experimental data
 - » Subsonic flow throughout
 - *Case 2:* $\alpha = 12^\circ$, $\delta = 20^\circ$, $M_\infty = 0.485$, flap gap = 0.15 inches
 - » Separated flow over flap
 - » Supersonic region near nose of wing
- Four different levels of physics: Euler, laminar, fully turbulent w/one- and two-equation turbulence models
- Three levels of geometric complexity: 2D airfoil/flap, 3D wing/flap/plates, 3D wing/flap/plates/mount/tunnel walls

The lack of agreement in 2D was not unexpected, as the flap was only partial span. Euler completely misses the trailing edge of the flap, laminar is unsteady, and the two turbulent solutions have unexpectedly separated partway along the flap.



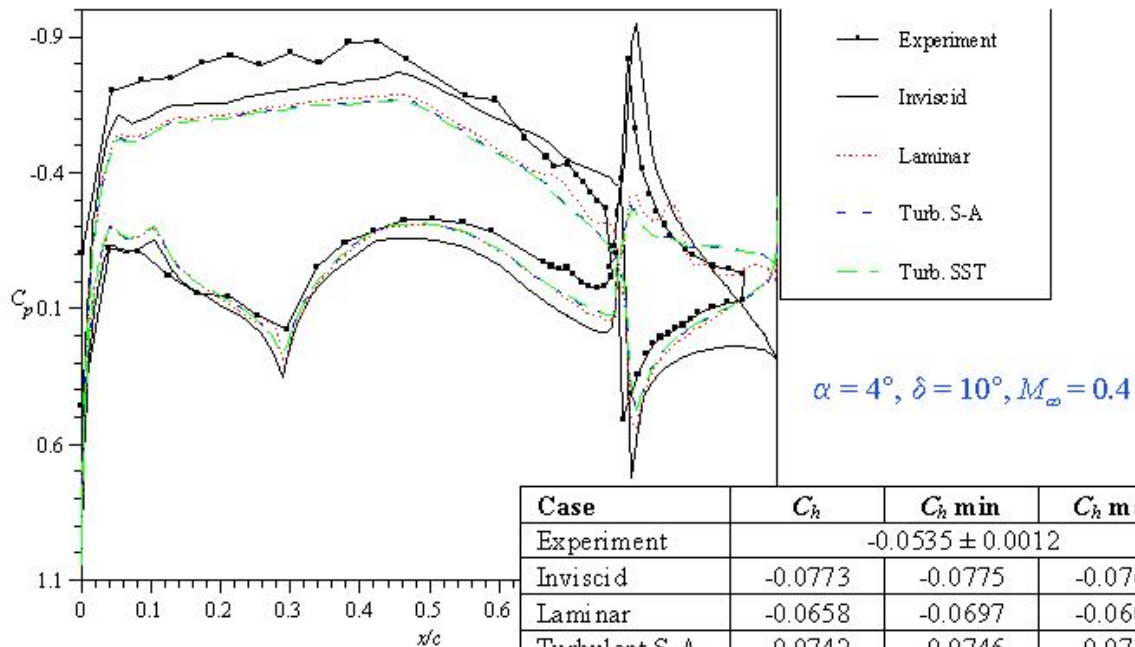
Case 1 : Two-Dimensional



Going the 3D really brings the solutions much closer to the experimental data. Unfortunately the separation on the flap is still evident in the turbulent solutions. Hinge moment results are much improved over the 2D solution, but still off by about 50%.



Case 1: 3D w/o Tunnel Walls

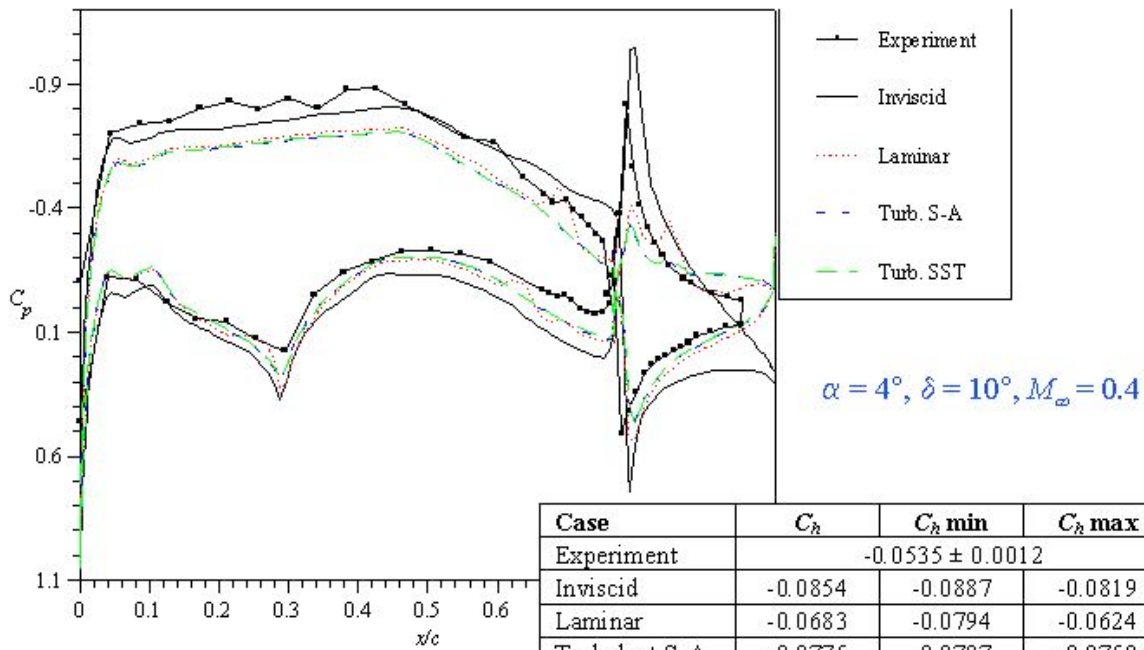


Case	C_h	$C_h \text{ min}$	$C_h \text{ max}$
Experiment	-0.0535 ± 0.0012		
Inviscid	-0.0773	-0.0775	-0.0767
Laminar	-0.0658	-0.0697	-0.0603
Turbulent S-A	-0.0742	-0.0746	-0.0739
Turbulent SST	-0.0725	-0.0729	-0.0722

Including the tunnel walls improved the solutions, resulting in a good match along the lower surface of the wing for the viscous solutions. All the solutions moved closer to the experimental data along the upper surface of the wing. The turbulent solutions are still separated on the flap, and laminar is still unsteady.



Case 1: 3D w/Tunnel Walls

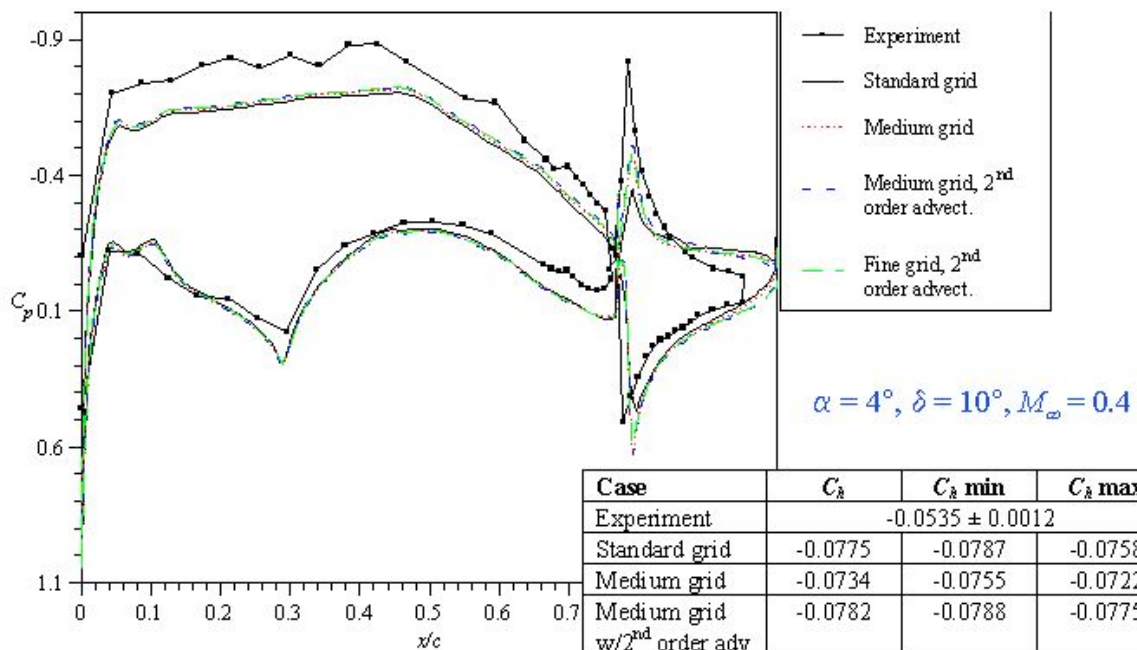


Case	C_h	$C_h \text{ min}$	$C_h \text{ max}$
Experiment		-0.0535 ± 0.0012	
Inviscid	-0.0854	-0.0887	-0.0819
Laminar	-0.0683	-0.0794	-0.0624
Turbulent S-A	-0.0775	-0.0787	-0.0758
Turbulent SST	-0.0768	-0.0771	-0.0765

The medium grid with 2nd order advection for the turbulence quantities is the best match to the experimental results.



Case 1: 3D w/Walls, Increased Resolution and Turb. Accuracy

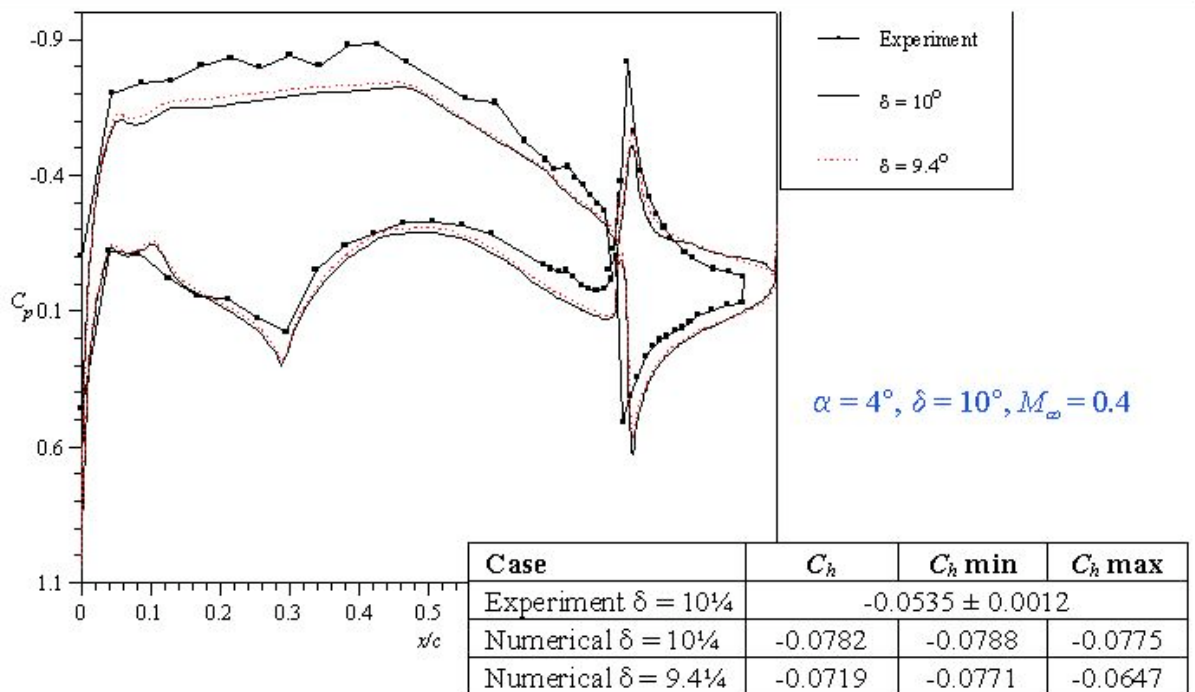


Case	C_h	C_h min	C_h max
Experiment	-0.0535 ± 0.0012		
Standard grid	-0.0775	-0.0787	-0.0758
Medium grid	-0.0734	-0.0755	-0.0722
Medium grid w/2 nd order adv	-0.0782	-0.0788	-0.0775
Fine grid w/2 nd order adv.	-0.0805	-0.0815	-0.0794

Considering the uncertainty in the flap angle due play in the flap and flexing in the moment-measuring torsion cell leads to better agreement with the moment measurements.



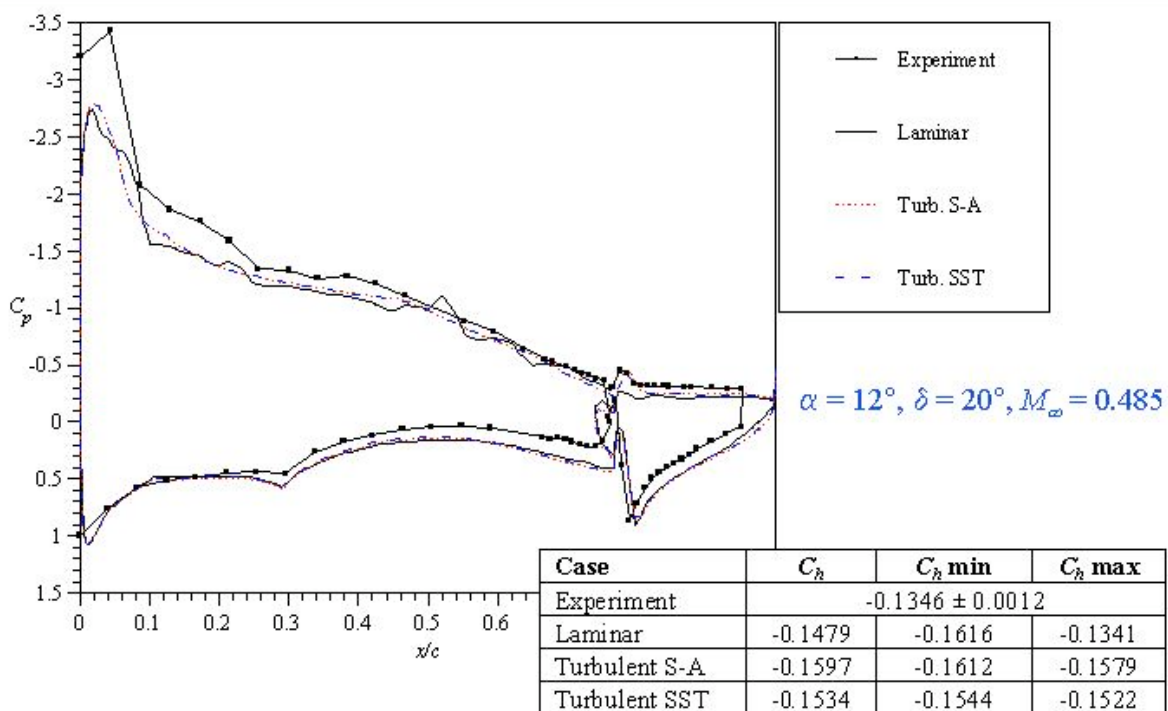
Case 1: 3D w/Walls, Uncertainty



Because there is now separation in both the experimental and numerical data, the hinge moment agreement between the two is now much better. Numerical solutions are still missing the suction peak in the experimental data, suggesting there may still be a problem with the digital representation of the model at the leading edge.



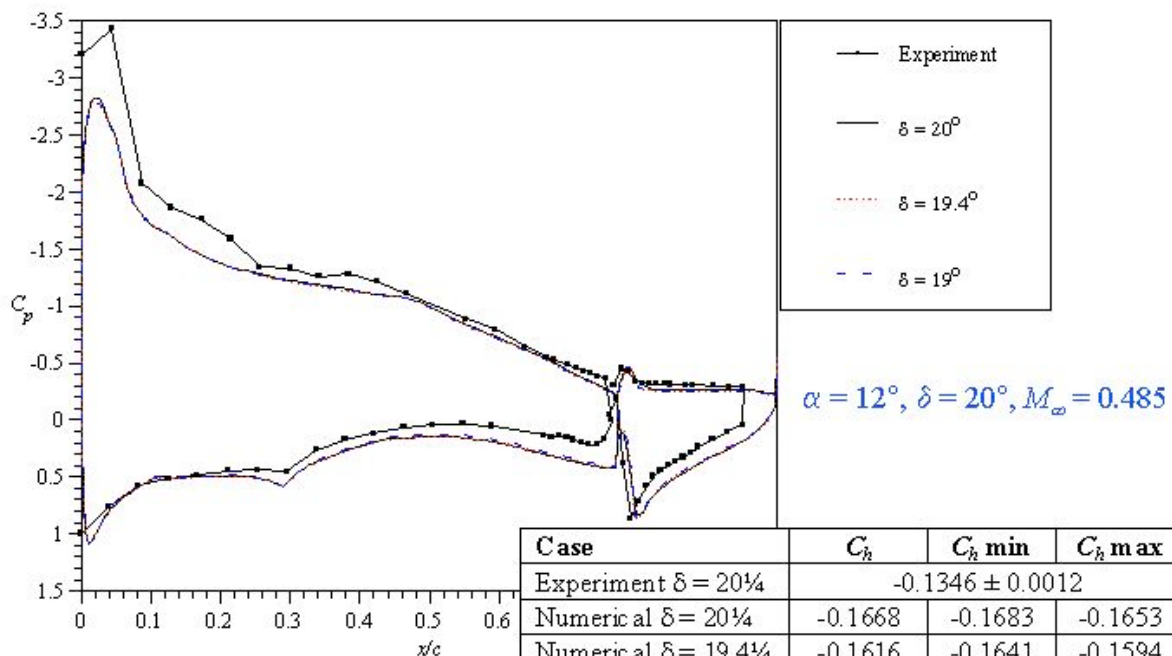
Case 2: 3D w/Tunnel Walls



Considering the uncertainty in the flap, in this case broken up into the flex in the torsion cell and the play in the flap, lead to very little difference in the pressure coefficients. The hinge moment does indicate a slight improvement.



Case 2: 3D w/Walls, Uncertainty

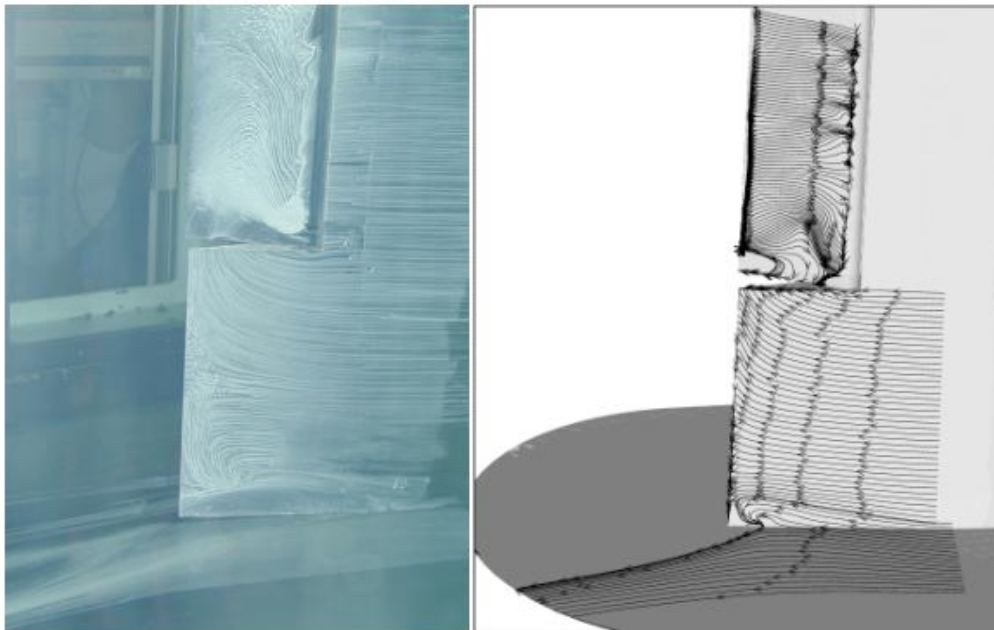


Case	C_h	C_h min	C_h max
Experiment $\delta = 20^\circ$	-0.1346 ± 0.0012		
Numerical $\delta = 20^\circ$	-0.1668	-0.1683	-0.1653
Numerical $\delta = 19.4^\circ$	-0.1616	-0.1641	-0.1594
Numerical $\delta = 19^\circ$	-0.1595	-0.1609	-0.1582

Qualitatively the computed solutions are very similar to the wind tunnel data, showing the same types of features as the oil flow on the model.



Surface Oil Flow Comparison



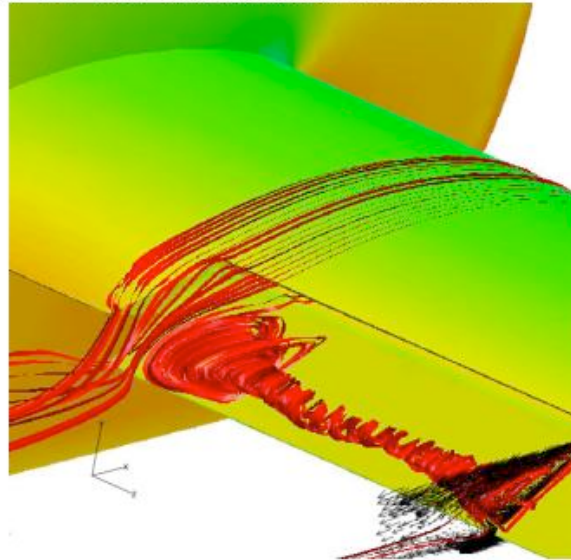
$$\alpha = 12^\circ, \delta = 19.4^\circ, M_\infty = 0.485$$

Separation on the flap is evident in the streamlines and velocity vectors of the computed solution.



Numerical Flow Visualization

- Surface pressure coefficient, velocity vectors and streamlines for 3D numerical solution



$$\alpha = 12^\circ, \delta = 19.4^\circ, M_\infty = 0.485$$

We learned after the test that there were also maintenance issues with the tunnel. A number of the screens in the inlet of the tunnel are torn or have fallen down, leading to nonuniform flow in the test section.



Conclusions

- **Experimental results most useful when test articles are mathematically defined and experimental uncertainties included**
- **Inviscid solutions cannot provide accurate hinge moments due to important viscous effects near the trailing edge of the control surface**
- **Modeling wind tunnel walls produces significant effects in numerical solutions, and may be necessary when comparing against wind tunnel data**
- **Accurate prediction of flow separation is key to accurately determining hinge moments numerically**



Acknowledgements

- **The High Performance Computing resources of the following organizations were instrumental in completing this work**
 - **U. S. Army High Performance Computing Research Center (AHPCRC), Minneapolis, Minnesota**
 - **U. S. Army Engineer Research and Development Center Major Shared Resource Center (ERDC MSRC), Vicksburg, Mississippi**
 - **U. S. Air Force Aeronautical Systems Center Major Shared Resource Center (ASC MSRC), Dayton, Ohio**
- **Special thanks to W. Strang for his continual advice and support over the course of this study**

CFD Simulation of Aircraft in Coning Motion

Syta Saephan and C.P. (Case) van Dam

Department of Mechanical and Aeronautical Engineering
University of California, Davis

September 24, 2003

NASA Symposium on
Computational Methods for Stability and Control

UCDAVIS

Acknowledgements

Mike Fremaux

Vehicle Dynamics Branch, NASA Langley Research Center

Teryn DalBello

NASA Glenn Research Center

Roger Strawn, Mark Potsdam

Army/NASA Rotorcraft Division, NASA Ames Research Center

William Chan, Tor Nygaard

ELORET Corp., NASA Ames Research Center

UCDAVIS

This presentation is structured in a timeline format. Some introductory material and sample results for this project are presented. Scattered throughout the presentation are some “lessons learned” slides which convey insights that may be of some usefulness to the attendees of this symposium.

Outline

- **Introduction and motivation**
- **Project goals and timeline**
- **Computational model and results**
- **Lessons learned**
- **Summary**

Accurate determination of the stability and control characteristics of an aircraft is critical to the safety of that aircraft. The flight envelope of some aircraft often pushes the limits on angle of attack and rapid motion maneuvers. At these flight conditions the flow about the aircraft tends to be unsteady and separated and the variation in the forces and moments with angle of attack or motion rate are often nonlinear.

Introduction

- **Advanced aircraft are very agile with flight envelopes that include high angles of attack and dynamic conditions**
- **Massively separated vortical flows**



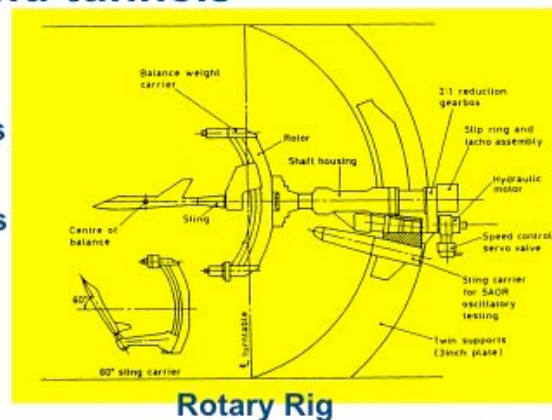
UC DAVIS

During the development process of advanced aircraft, numerous hours are spent testing scaled models in wind tunnels to determine the stability and control characteristics. Much of this testing is conducted using models of simplified geometry at Reynolds numbers much lower than typical flight Reynolds number. However, the sensitivity of the forces and moments to Reynolds number makes these empirical approaches to predict dynamic characteristics at full scale high angle of attack conditions a challenging task.

Much of the wind tunnel testing is conducted with special equipment designed for this purpose. Rotary balance apparatuses, similar to the one shown in the slide, were developed to provide information on the effects of angular rates on the aerodynamic forces and moments acting on the

Motivation

- **Stability and control characteristics under these conditions are typically predicted through wind tunnel tests**
- **Disadvantages of wind tunnels**
 - ♣ Complex
 - ♣ Expensive
 - ♣ Few dynamic testing tunnels
 - ♣ Interference effects
 - ♣ Reynolds number limitations



UCDAVIS

aircraft in flight. The apparatuses are typically complex because they must be capable of measuring forces and moments for the range of rotation rates experienced by high-performance aircraft. Major problems encountered in the application of this test technique include test equipment interference, wind tunnel wall effects, equipment blockage ratio, and Reynolds number scaling effects in addition to the difficulty of conducting detailed flow measurements such as surface pressures.

This study is motivated by the need of industry to quickly and inexpensively determine stability and control characteristics of new aircraft without solely having to resort to wind tunnel experiments. In particular, rotary balance testing at high Reynolds numbers is complex because of the high structural loads imposed on the model and the rotary test apparatus. The high loading is the result of the combined effects of high dynamic pressure of the wind tunnel flow and the high angular velocities required to match the flow velocities for a given spin coefficient. The potentially cheaper and faster aspects of CFD make it a helpful addition to experiments, if it can be proven to be reliable under dynamic conditions.

Project Goals

- **Use CFD to simulate generic aircraft forebodies at high angles of attack rotary conditions**
- **Compare CFD results to wind tunnel experiments**
- **Simulate a complete aircraft configuration under similar conditions**

UCDAVIS

This project began prior to year 2000 with Teryn DalBello and Case van Dam. The computational grids were created from physical measurements of the actual test models. Some preliminary results were computed on NAS' CRAY machines using the rotorcraft version of Overflow. Syta Saephan joined the project in year 2000 and continued running some additional cases.

Project Timeline

- ➔ **2000 – Begin forebody flow simulation using rotorcraft version of Overflow on NAS' CRAY machines**
- 2001 – Compare CFD results to available wind tunnel data
- 2002 – Transition to Overflow-D, chimera grid approach, dynamic /time accurate simulations
- 2003 – Start F16XL project using Overflow-D on network of Linux PCs

UCDAVIS

The ultimate goal of the project was to assess the capability of CFD (and Overflow in particular) in predicting the flow about aircraft forebodies at high angle of attack rotary conditions. The plan was to start off with something simple and manageable to gain experience and familiarity with the grid generation tools and flow solver. The ogive geometry was chosen for two reasons. First, the geometry resembled that of advanced aircraft and would produce similar vortical flow structures. Second, experimental data are available to validate the computational results.

Forebody Simulation

- **Simple geometry, reduces possible issues involving geometry/grid**
- **Wind tunnel data available for comparison**



UCDAVIS

In the mid 1990s, rotary balance experiments were conducted in Britain on isolated circular and square cross section ogive models at angles of attack of 60° and 90° over a range of Reynolds numbers from 80,000 to 2,250,000 based on the maximum body diameter.[1],[2] The purpose of these experiments was to determine the effects of Reynolds number, angular velocity, and nose shape on the aerodynamic characteristics of the models. These tests were unique in so far that this was the first time that surface pressure distributions were measured under rotary conditions in a pressurized wind tunnel. A second objective of these experiments was to provide a database for the development and validation of high angle of attack computational methods. This database forms the basis for the computational investigations of this report.

Experimental Data

- Conducted in mid 1990s by NASA/DRA
- Measured forces, moments, and surface pressures for circular and rectangular ogives
- Parameters:
 - $\alpha = 60^\circ, 90^\circ$
 - $M_\infty = 0.024 \text{ _ } 0.21$
 - $Re_D \approx 80,000 \text{ _ } 2,250,000$
 - $b/(2V_\infty) = 0 \text{ _ } \pm 0.4$

UCDAVIS

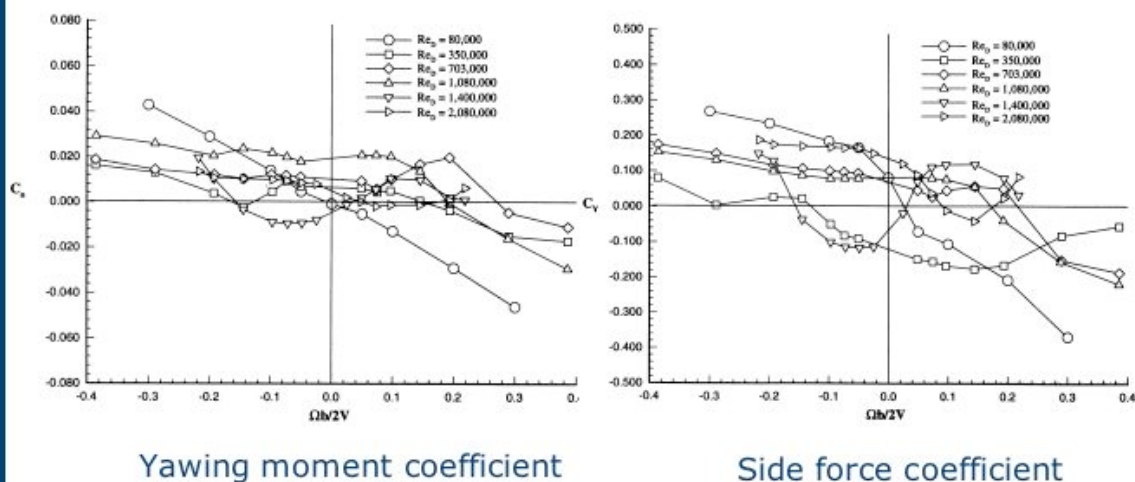
[1] Dunham, D.M., "Rotary Results on Forebody Models," Cooperative Programme on Dynamic Wind Tunnel Experiments for Maneuvering Aircraft, AGARD AR-305, October 1996, pp.7-1-7-14.

[2] Pauley, H., Ralston, J., and Dickes, E., "Experimental Study of the Effects of Reynolds Number on High Angle of Attack Aerodynamic Characteristics of Forebodies During Rotary Motion," NASA CR 195033, January 1995.

These two plots show the side force and yawing moment coefficients as a function of spin rate at various Reynolds numbers as measured in a spin tunnel. Notice that the forces and moments are highly nonlinear, even changing signs, over the range of spin rates and Reynolds numbers.

Sample Experimental Data Set

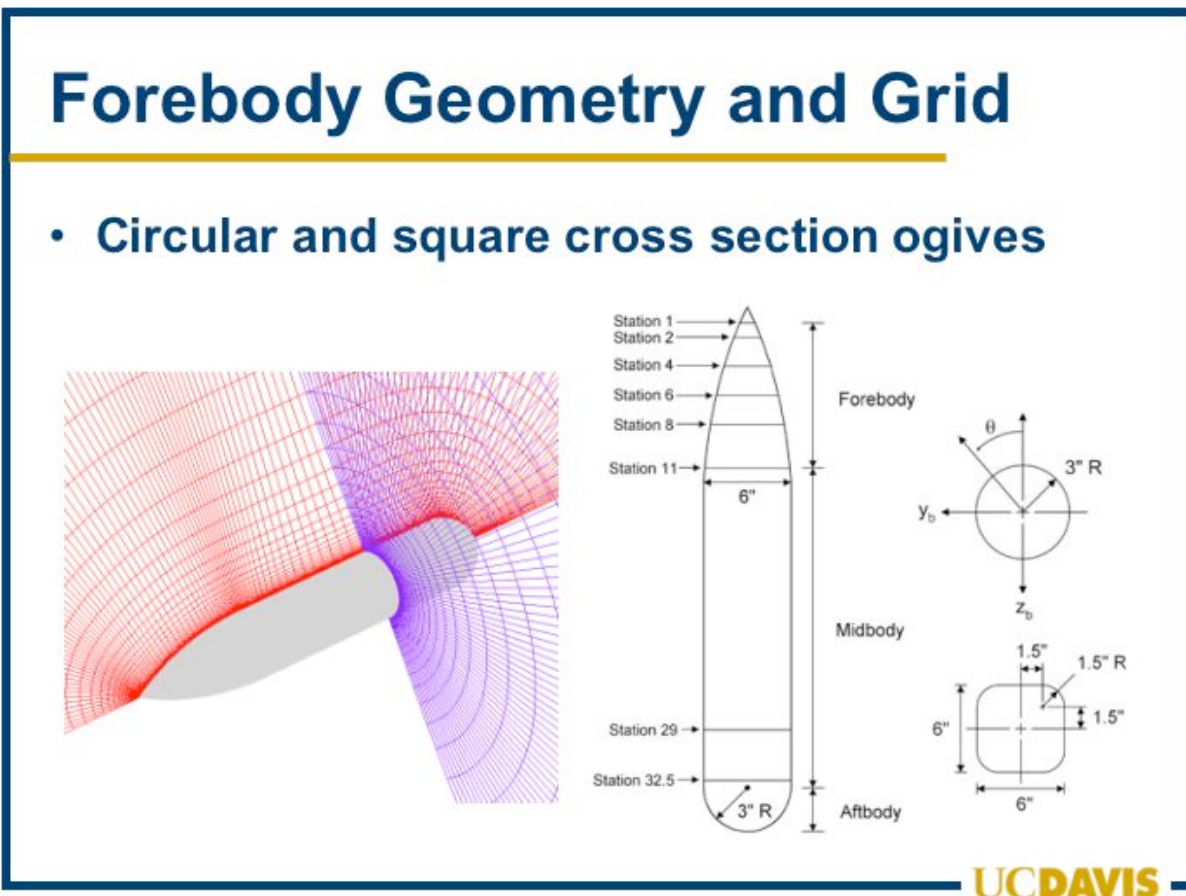
- Circular ogive over a range of Reynolds numbers



UCDAVIS

The computational grids were created from detailed physical measurements of the test models. Several geometries were used in the wind tunnel experiments, but only two geometries are used for this project. Both ogives are 36 inches in length and 6 inches in diameter. The difference between the two geometries is their cross sectional shape, with one being circular and the other being a square with rounded corners. Both computational grids are single grids with 130 points distributed in the axial direction, 181 points in the circumferential direction, and 54 points in the normal direction.

Surface pressure data were measured with pressure taps on the test models. The measurements were taken at the axial locations labeled as stations on the slide. The station number refers to its location from the nosecone tip. Station 1 is one inch from the nosecone tip. Station 29 is 29



inches from the nosecone tip. All stations had 32 equally spaced pressure taps except for Station 1 which only accommodated 30 taps.

Note that the ogive is divided into three regions, the forebody, midbody, and aftbody. The forebody region is bounded by Stations 1 and 11 and will be referenced as “forebody”. The complete configuration comprising of all three regions will be referenced as “ogive”.

The flow solver used for this project is OVERFLOW. There are several versions of OVERFLOW in circulation, each having been specially modified to serve a particular need. We have used different versions of OVERFLOW for this project. The results shown in this presentation were obtained using the rotorcraft version of OVERFLOW 1.6.

Despite the numerous versions afloat, all derivative versions of OVERFLOW share some commonality. OVERFLOW is a Reynolds averaged Navier Stokes flow solver for structured grids. It can solve problems on single or chimera overset grids, with the more recent versions having more overset grid capability. Users can choose which turbulence model to use from a selection that includes Baldwin-Lomax, Baldwin-Barth, Spalart-Allmaras, and other higher order models as well.

Flow Solver

- **OVERFLOW**
- **Structured RANS solver**
- **Single or overset grids**
- **Turbulence models:**
 - Baldwin-Lomax, Spalart-Allmaras, etc.
- **Static grid with rotational source terms**
- **Dynamic grid for rotational motion**

UC DAVIS

One feature that is incorporated into the rotorcraft version of OVERFLOW is the use of source terms for constant rotary motion. The source terms allow the simulation of rotary motion with a static grid. Rotary motion can also be simulated by physically rotating the grid at each timestep in a time accurate mode. In this case, the source terms would not be needed.

The simulation parameters for the result presented in this presentation are shown in the slide. The Reynolds number based on the ogive diameter is just over 2 million and the flow is assumed to be fully turbulent. These cases represent the highest Reynolds number cases tested in the wind tunnel. Three different spin coefficients were tested for each geometry. The spin coefficient is proportional to the ratio of the angular velocity and the freestream velocity.

The cases will be simulated using a static grid and source terms for the rotational effects. Baldwin-Lomax algebraic turbulence model is used because other researchers have achieved better results with this model in these high-alpha flow problems.

CFD Simulation Parameters

- **Static grid with rotational source terms**
- **$\alpha = 60^\circ$**
- **$M_\infty = 0.21$**
- **$Re_D \approx 2,000,000$ (assumed fully turbulent)**
- **$\omega/(2V_\infty) = 0.0, -0.1, -0.2$**
- **Baldwin-Lomax turbulence model**

UCDAVIS

Numerous cases were computed on the CRAY machines. The results need to be validated against the wind tunnel data.

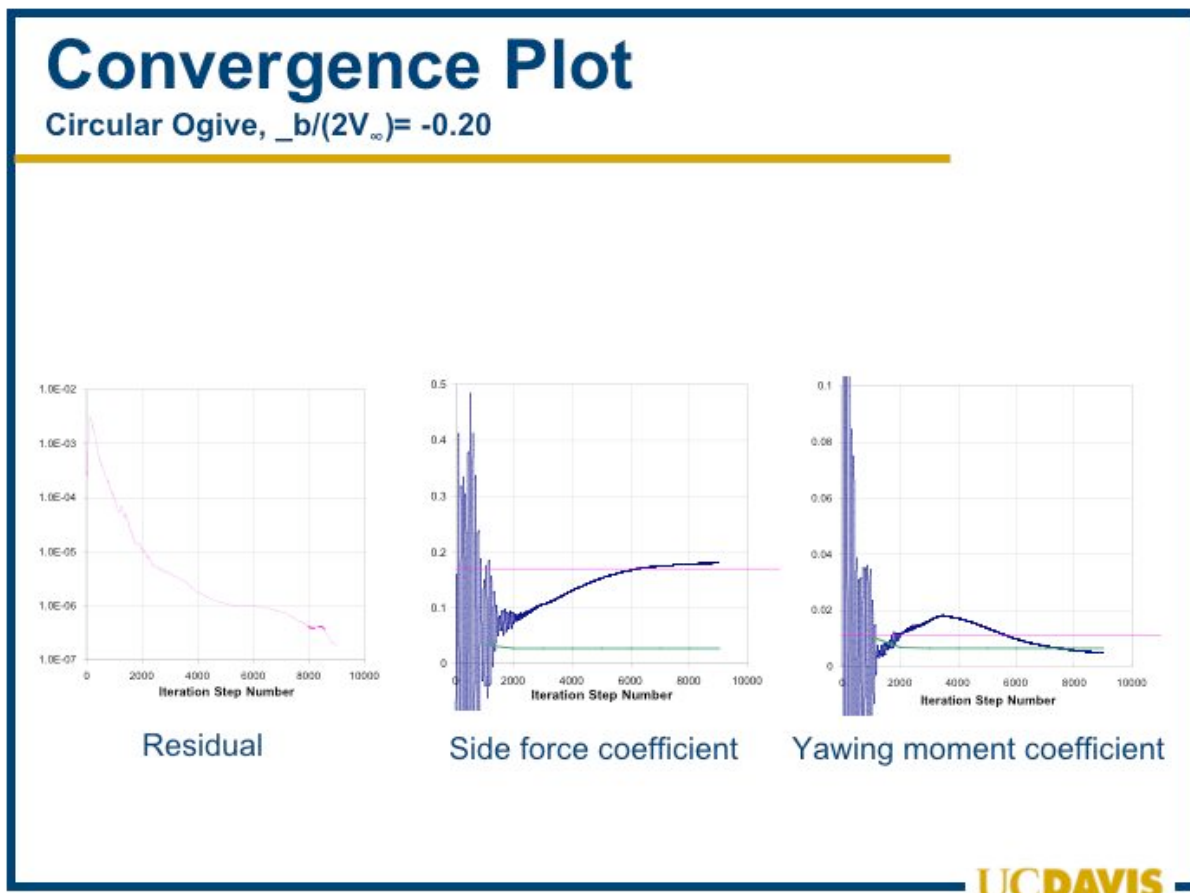
The experiments measured the side force and yawing moment acting on the model via a vertically mounted sting at the model's center of gravity. Surface pressures were also measured at the six forebody stations and two aftbody stations shown in an earlier slide. The surface pressure measurement allowed for a more detail comparison of the forces and moments acting on the forebody region of the model. The measured surface pressures are integrated over the forebody region to obtain the forces and moments acting on the forebody region.

Project Timeline

- 2000 – Begin forebody flow simulation using rotorcraft version of Overflow 1.6 on NAS' CRAY machines
- ➔ **2001 – Compare CFD results to available wind tunnel data**
- 2002 – Transition to Overflow-D, chimera grid approach, dynamic/time accurate simulations
- 2003 – Start F16XL project using Overflow-D on network of Linux PCs

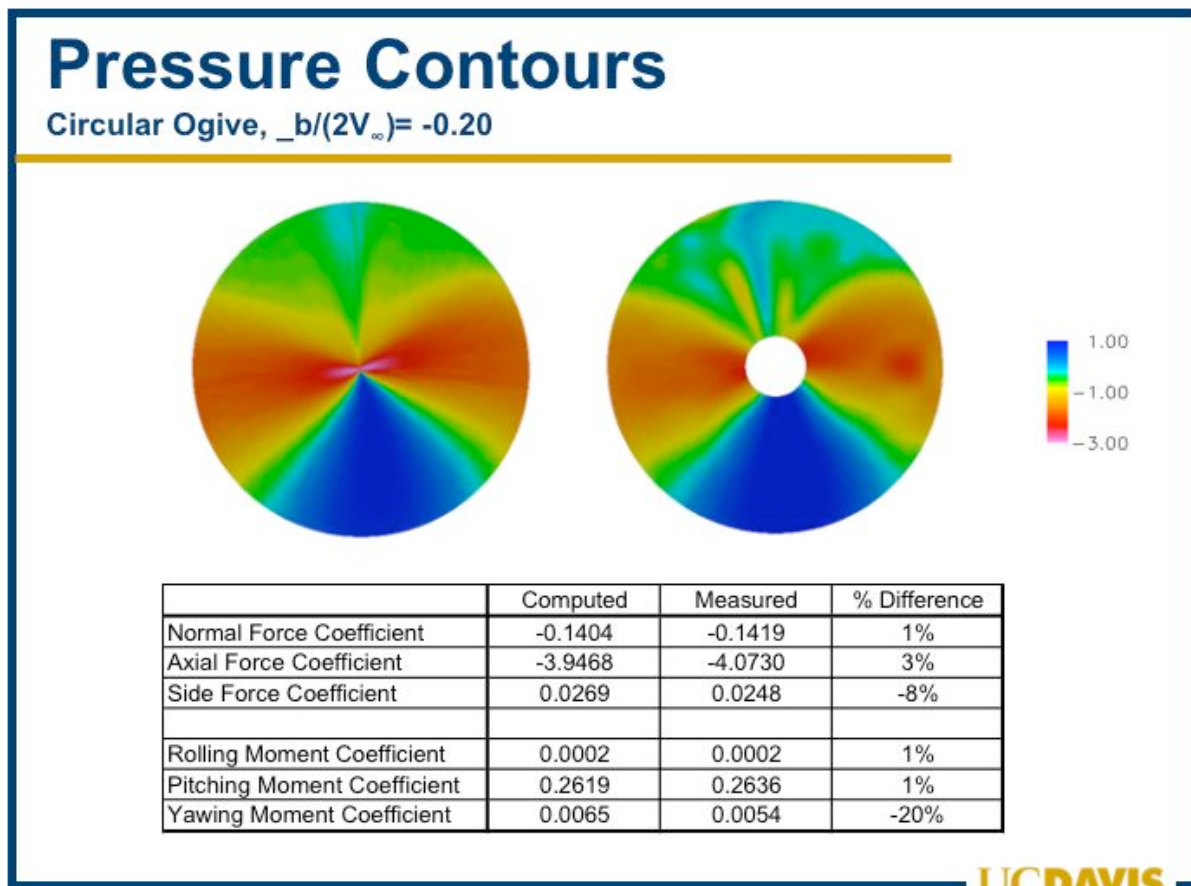
UCDAVIS

These three plots show the drop in the residual and the convergence of the side force and yawing moment. The residual in terms of the L2-norm is reduced by three orders of magnitude and is approaching machine zero. The side force for the ogive (shown in blue) starts out very oscillatory but does dampen out and does converge in less than 10,000 iterations for this case. The side force for the forebody region (shown in green) converges much faster than the overall ogive side force. In fact, the forebody forces and moment typically converge 60-80% faster. A similar convergence pattern is seen for the yawing moment.



The color contours compare the computational (left) and experimental (right) surface pressure distribution for the circular ogive case at a spin coefficient of -0.2. The experiments did not make any measurements at the very tip of the forebody and hence the white hole in the middle of the picture. The blue region is on the windward side.

Qualitatively, the comparison is not bad with the skewness and gradient patterns being captured in the simulation. To compare the results on a more quantitative level, the surface pressure is integrated to get the forces and moments acting on the forebody. The results of the surface pressure integration is shown in the table. The normal and axial components of the force along with the rolling and pitching components of the moment agreed to within a few percent of the

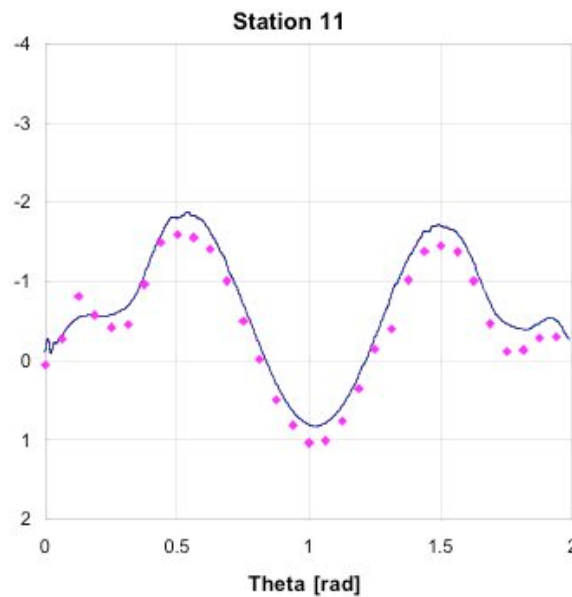
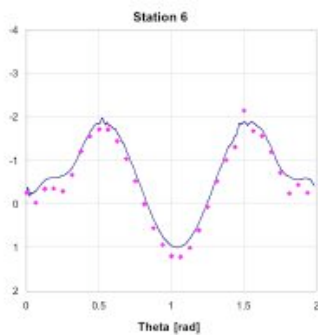
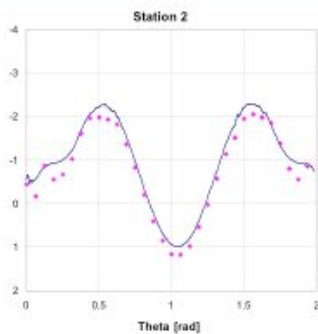


experimental values. The side force and yawing moment components are the more difficult components to predict and the discrepancy is larger.

Whereas the last slide compared the surface pressure distribution in a colorful and qualitative manner, these pressure plots compare the results in a more quantitative manner. These three plots show the pressure distribution on the forebody at stations 2, 6, and 11. The trend predicted by the flow solver agrees with that measured in the tunnel, although pressures on the windward side are predicted with a higher degree of accuracy than values on the leeward side. The computed pressures are slightly yet noticeably shifted upwards when compared to the measured values. This small shift may be attributed to the known reference pressure error in the experimental data.

Pressure Plots

Circular Ogive, $_b/(2V_\infty) = -0.20$

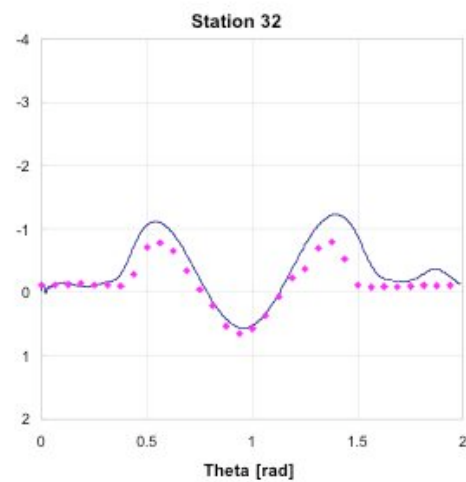
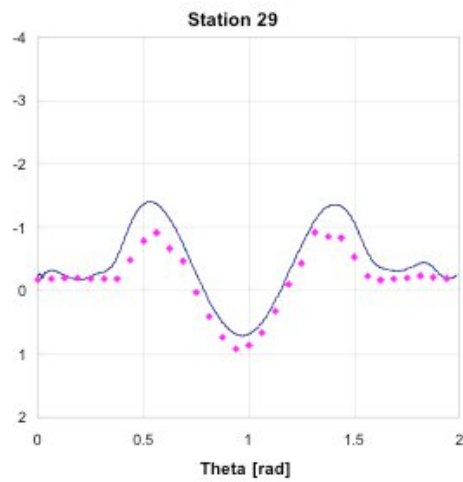


UCDAVIS

These two plots compare the surface pressures at the two stations on the aftbody. Again, the solver captures the trend of the surface pressure.

Pressure Plots

Circular Ogive, $_b/(2V_\infty) = -0.20$



UCDAVIS

The circular ogive geometry was simulated at three spin coefficients. Rather than show line plots and color contours for each case, the results have been consolidated into this table comparing forebody forces and moments. Except for the side force and yawing moment, all of the other components have less than a 5% error relative to the experimental results. The errors in the side forces are generally smaller than errors in yawing moments as expected. Moments are much more difficult to predict as they are more sensitive to variations in surface pressure far from the moment center.

Circular Ogive Results

	CFD			Experiment		
$\Omega b/2V_\infty$	-0.2	-0.1	0.0	-0.2	-0.1	0.0
C_N	-0.1404	-0.1400	-0.1404	-0.1419	-0.1411	-0.1414
C_A	-3.9468	-3.9478	-3.9480	-4.0730	-4.0717	-4.0710
C_Y	0.0269	0.0166	0.0089	0.0248	0.0193	0.0139
C_l	0.0002	0.0002	0.0002	0.0002	0.0002	0.0002
C_m	0.2619	0.2612	0.2618	0.2636	0.2624	0.2625
C_n	0.0065	0.0035	0.0011	0.0054	0.0037	0.0020

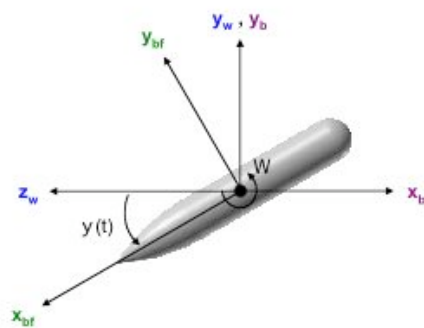
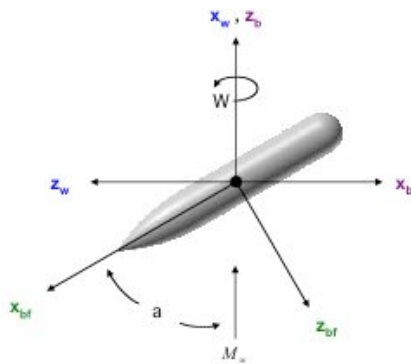
UCDAVIS

No one really gives reference frames a second thought and this is okay 90% of the time when all the different frames are aligned and identical. However, there may be instances where multiple reference frames exist and one needs to pay special attention to make sure all comparisons are with respect to the same reference frame.

For our problem, there are three different reference frames in use. The wind tunnel experiments measured forces and moments in body-fixed axes with the x-axis aligned with the axis of the model. Ideally, the CFD computational domain should employ the same axis orientation for a simple and direct comparison of results. However, there may be additional constraints imposed by the gridding process or flow solver that dictate a certain axis orientation as was the situation

Lessons Learned!

- **Beware of different reference frames when comparing forces/moments**
 - Stability axes, body axes, body-fixed axes
 - Most likely need one transformation for “static grid” simulations and two transformations for “dynamic grid” simulations



UCDAVIS

for this project. The source term formulation restricts the rotary motion about the z-axis only. This single restriction led to the use of three separate reference frames. As mentioned, the experimental data referenced the body-fixed system. OVERFLOW reported forces in the wind axis system and moments in the body axis system. To directly compare the computational results with the experimental data, all computational forces and moments will be transformed into the body-fixed axes system.

Another lesson learned is that the forebody region is much more accurately predicted by the solver than the ogive as a whole. One possible reason why the forebody results tend to be better than ogive results is that the sting is not modeled in the computations. The forebody is upstream of the vertically mounted test model sting and the flow in that region is less affected by the sting. Flow in regions downstream of the sting are undoubtedly affected by the sting. Hence, the lack of the sting in the computational model means that its effects are not captured and is a reason for the ogive result discrepancy.

Also, other wind tunnel interferences such as tunnel wall and the presence of the rotary rig (none of which are modeled in the simulations) can affect the solution as well.

Lessons Learned!

- **Good forebody comparison can still lead to bad comparison of wind tunnel balance measurements**
 - Sting effects?
 - Other wind tunnel interferences?
 - Or just a CFD shortcoming?

Coefficients	Forebody		Complete Body	
	CFD (Pressure Integration)	Experiment (Pressure Integration)	CFD (Pressure & Skin Friction Integration)	Experiment (Balance Measurement)
Normal Force	-0.14001	-0.14113	-0.31179	
Axial Force	-3.94782	-4.07170	0.05984	
Side Force	0.01660	0.01935	0.05214	0.160
Rolling Moment	0.00016	0.00016	0.00017	
Pitching Moment	0.26123	0.26236	0.27327	
Yawing Moment	0.00346	0.00373	0.01298	0.010

$$C_b / (2V_\infty) = -0.10$$

UCDAVIS

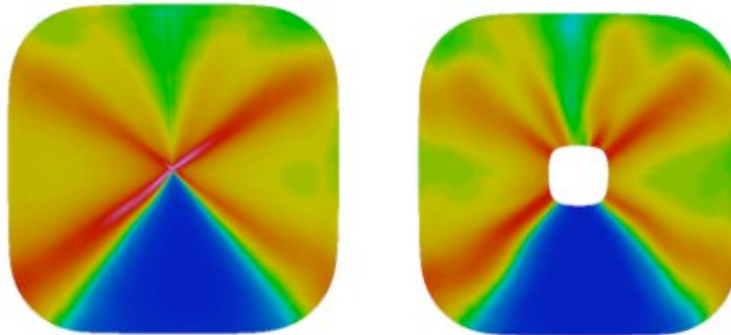
Can the ogive force and moment discrepancy be completely blamed on wind tunnel interferences or are there other culprits?

The color contours compare the computational (left) and experimental (right) surface pressure distribution for the square ogive case at a spin coefficient of -0.1. The scale for the surface pressure is the same as that of the circular ogive case.

Qualitatively, the comparison is fair with the major flow features captured. Quantitatively, the computed forebody forces and moments differ the experimental values by varying amounts. The normal force, axial force, and pitching moment agree with the experimental values to within 11%. However, the other components of forces and moments differ by significant percentages. It should be noted that some of these force and moment coefficients are very small and hard to predict and even small differences in value can result in large percentage deviations.

Pressure Contours

Square Ogive, $\alpha/(2V_\infty) = -0.10$



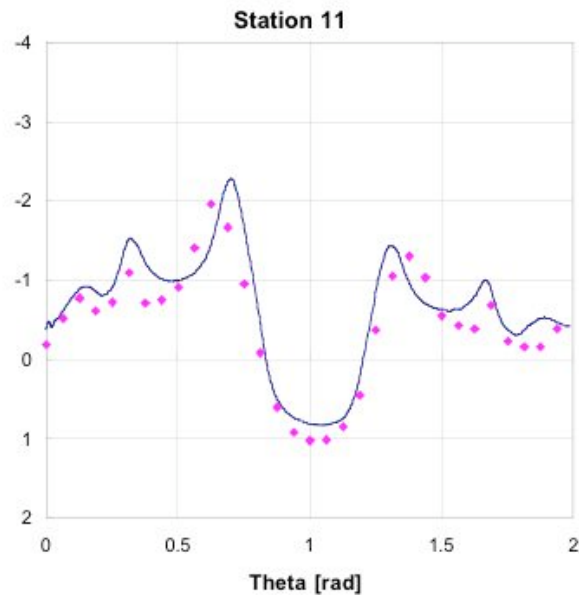
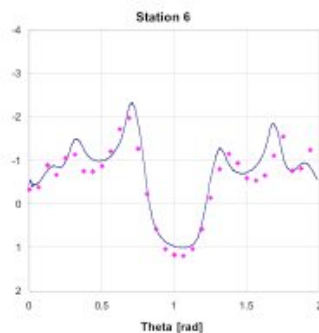
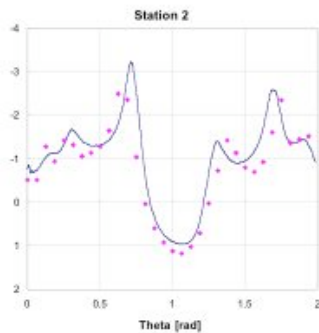
	Computed	Measured	% Difference
Normal Force Coefficient	-0.1913	-0.2142	11%
Axial Force Coefficient	-4.7460	-4.7701	1%
Side Force Coefficient	0.0411	0.0302	-36%
Rolling Moment Coefficient	-0.0001	0.0000	-237%
Pitching Moment Coefficient	0.3578	0.4019	11%
Yawing Moment Coefficient	0.0113	0.0080	-41%

UCDAVIS

These three plots show the pressure distribution on the square ogive's forebody section at stations 2, 6, and 11. The flow solver predicts the trend of the surface pressure although there are some discrepancies when comparing absolute values. The pressures on the windward side are predicted with a higher degree of accuracy than values on the leeward side, as was the case for the circular ogive geometry.

Pressure Plots

Square Ogive, $\alpha/(2V_\infty) = -0.10$

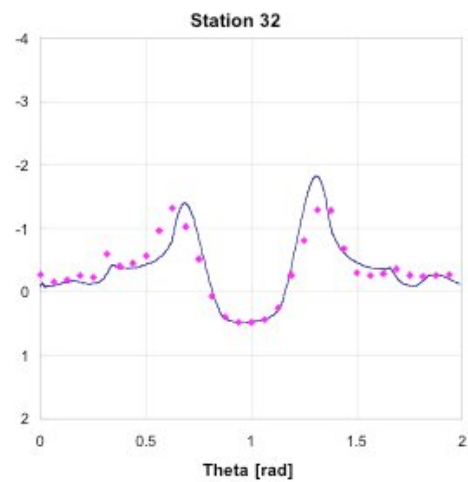
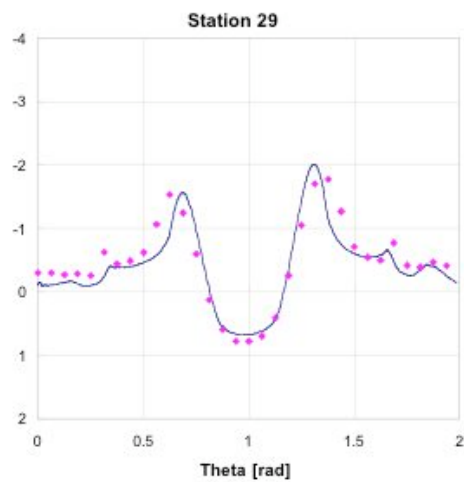


UCDAVIS

These two plots compare the surface pressures at the two stations on the aftbody.

Pressure Plots

Square Ogive, $_b/(2V_\infty) = -0.10$



UCDAVIS

The square ogive geometry was simulated at three spin coefficients. This table compares the predicted forebody forces and moments to the experimental values. Except for the side force and yawing moment, all of the other components have less than a 5% error relative to the experimental results. The errors in the side forces are generally smaller than errors in yawing moments as expected. Moments are much more difficult to predict as they are more sensitive to variations in surface pressure far from the moment center.

This table (and the one for the circular ogive) show nonzero side force, rolling moment, and yawing moment coefficients for the case with zero rotation. There are several reasons why they should not be zero as confirmed by the nonzero values measured in the wind tunnels. Neither of

Square Ogive Results

	CFD			Experiment		
$\Omega b/2V_\infty$	-0.2	-0.1	0.0	-0.2	-0.1	0.0
C_N	-0.1904	-0.1913	-0.1917	-0.2166	-0.2142	-0.2134
C_A	-4.7448	-4.7460	-4.7462	-4.7797	-4.7701	-4.7569
C_Y	0.0786	0.0411	0.0040	0.0564	0.0302	0.0039
C_l	-0.0002	-0.0001	0.0000	-0.0001	0.0000	0.0000
C_m	0.3568	0.3578	0.3585	0.4078	0.4019	0.3998
C_n	0.0219	0.0113	0.0009	0.0156	0.0080	0.0007

UCDAVIS

the two wind tunnel models is perfectly symmetric and the slight asymmetry is replicated in the surface grids. Even for nominally symmetric bodies the flow tends to become asymmetric in the angle of attack range from approximately 20° to 70° . At lower angles of attack, the flow remains symmetric whereas at higher angles unsteady vortex shedding occurs. Hence, it is not unexpected that the side force, rolling moment, and yawing moment are nonzero at the stationary, condition for the bodies considered here.

By year 2002, NAS has replaced its older CRAY machine with a newer one. This newer CRAY will be operational for one to two years, after which time NAS will no longer use CRAYs. Rather than transition over to the new CRAY, this was an appropriate time to transition to parallel computing (on NAS' SGI clusters) and our own Linux clusters. The transition to new hardware was also an appropriate time to transition to a newer version of OVERFLOW. OVERFLOW-D has major improvements over its predecessor. OVERFLOW-D is ideal for chimera overset grids with a built-in ability to fill the computational domain with Cartesian volume grids. Also, the flow solver can better load balance the problem between parallel processors by splitting large grids into smaller grids. OVERFLOW-D retained the source term capability, but has improved dynamic grid capabilities.

Project Timeline

- 2000 – Begin forebody flow simulation using rotorcraft version of Overflow 1.6 on NAS' CRAY machines
- 2001 – Compare CFD results to available wind tunnel data
- ➔ **2002 – Transition to Overflow-D, chimera grid approach, dynamic/time accurate simulations**
- 2003 – Start F16XL project using Overflow-D on network of Linux PCs

UCDAVIS

Several lessons were learned when we transitioned to OVERFLOW-D. We restricted ourselves to single grid topologies when we used the older rotorcraft version. When we transitioned to OVERFLOW-D, we tried to simulate the same case with overset grids. Even though the ogive geometry is very simple compared to aircraft geometries, we had difficulty making a good grid without any orphan points. We also ran into some problems with the hole cutting and domain connectivity steps. That problem was fixed with a small modification to a hole-cutting subroutine. Much of these problems and hardships can be avoided or minimized by using as few grids as possible.

An additional incentive to use single grid topologies whenever possible is that solutions on the a

Lessons Learned!!

- **Single grid topology is much simpler**
 - No orphan points
 - No need for hole cutting and domain connectivity step
 - Use as few grids/domains as possible!
- **Single grid simulations achieve steady state much faster than chimera overset grid simulations**
 - ~10,000 iterations for single grid
 - ~40,000+ iterations for chimera grids

UCDAVIS

single grid domain converges much faster than solutions on chimera grids domains. For the ogive geometry, single grid cases converge 75% faster than chimera grid cases.

These rotary problems can be simulated in two ways. The “fast” way is to simulate the problem with a static grid and use rotational source terms. The “slower” way is to simulate the problem with a dynamic grid that rotates at each time step. Because the grid is physically rotating, the source terms are not needed.

There are several things that need to be considered when deciding on which approach to take. First and foremost, can the problem be simulated with source terms. The use of source terms is only valid if the entire body is undergoing constant rotary motion and there is no relative motion between grid components. Secondly, does the flow solver support source term formulation?

Lessons Learned!!

- **Static grids simulations (with rotational source terms) are more desirable compared to dynamic/time accurate simulations**
 - Source terms still restrict rotation to z-axis
 - Will future Overflow versions (or other flow solvers) have that feature?
 - Time steps for time accurate simulations are incredibly small, on the order of 0.01° grid rotation per time step for flow conditions similar to that presented earlier
 - Subiterations (at least 3) are needed at each time step
 - Lots of additional flow solver overhead for dynamic and time accurate simulations

UCDAVIS

The use of source terms will result in significant computational resource savings. Dynamic simulations often require very small time steps for solution stability. For the ogive geometry and flow conditions described for this project, a time step equivalent to 0.01° grid rotation was needed for solution stability. At each time step, at least three subiterations are needed for proper solution development. For one complete grid revolution, the solution would have computed 100,000+ iterations.

At the current time, we are attempting to simulate the F16XL under coning motion.

Project Timeline

- 2000 – Begin forebody flow simulation using rotorcraft version of Overflow 1.6 on NAS' CRAY machines
- 2001 – Compare CFD results to available wind tunnel data
- 2002 – Transition to Overflow-D, chimera grid approach, dynamic/time accurate simulations
- ➔ **2003 – Start F16XL project using Overflow-D on network of Linux PCs**

UCDAVIS

We will be simulating an F16XL under coning motion. The first task is to create a CFD suitable grid from the CAD grid we obtained. That task will be difficult and time consuming for a complex geometry such as this. Once the gridding step is complete, we plan to simulate the problem on our homogenous network of Linux computers.

F16XL

- **Simulate an aircraft geometry (F16XL) at high angles of attack and rotary conditions**
- **Lots of gridding issues to transform a “CAD grid” to a CFD suitable grid**
- **Overflow-D using Linux cluster**



UCDAVIS

In recent years, computing power has increased significantly while prices have dropped just as significantly. We transitioned away from the CRAY and into parallel computing. Currently, NAS operates several massively parallel SGI machines with 512 processors and 1028 processors. Access to these machines is charged by the hour. Unless you need the massive computing power that these SGI computers provide, PC computer prices have dropped so much that it's economical to purchase a bunch of PCs and network them into a "mini" cluster.

We have assembled a homogenous network of 10 PCs running Linux. An Athlon XP 2500+ with 512MB of memory and a 80GB hard drive can be purchased for about \$500 each. These Linux clusters are highly scalable. The factor retarding further scalability (for us at least) is network

Lessons Learned!!

- **Price for computational resources have dropped dramatically**
- **Pay by the hour to use NAS's parallel SGI machines**
- **Use PCs with Linux OS, self assembled networks are relatively cheap and fairly scalable**
 - **Athlon XP 2500+ CPU, 512 MB RAM, 80GB HD (~\$500)**
 - **Communication between PCs appears to be the bottleneck for regular 100Mbit networks, 1000Mbit networking hardware available and becoming cheaper**

UCDAVIS

connection speed. We are using a 100 MBit ethernet cards and network switch. 1000 Mbit cards and switches are available but a bit expensive at the moment. They should become more affordable as they gain wider acceptance and a larger user base. With gigabit network connections, these Linux cluster can be scaled to any number of processors without any noticeable performance degradation.

We had two objectives in mind when putting together this presentation. First, we wanted to let people know what we were doing in terms of rotary flow problems. Ogives were simulated under coning motion to determine the feasibility and capability of OVERFLOW. Sample results were presented for cases simulated. The results for the forebody show reasonable agreement to experimental data.

Our second objective is to convey some of the lessons we learned during the course of the project. We hope that these lessons will be useful to the attendees of this workshop.

Summary

- **Circular and square ogives at 60° angle of attack and rotary motion have been simulated with a RANS flow solver**
- **CFD results show reasonable agreement to experimental data**
- **Many lessons learned along the way**

UCDAVIS



USE OF CFD-GENERATED AERODYNAMIC DATA FOR AN F-15E/SLV FLYING/HANDLING QUALITIES ANALYSIS

Jeffery Batte
Shawn Westmoreland
Jacobs Sverdrup
Eglin AFB, FL



Introduction



TASKING: AIR FORCE RESEARCH LABORATORY
(AFRL) REQUESTED AN ASSESSMENT BY THE
USAF SEEK EAGLE OFFICE FOR CARRIAGE OF THE
MICROSATELLITE LAUNCH VEHICLE (SLV)
ON AN F-15E AIRCRAFT



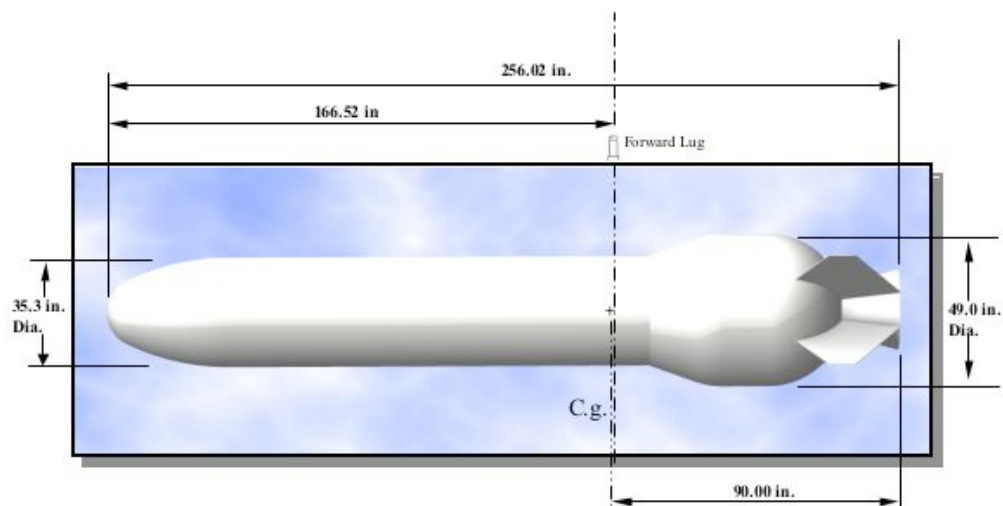
Introduction



- SLV store is larger than any other store carried on F-15E aircraft's centerline station , e.g., fuel tank, GBU-28, etc.
 - ♣ Carriage of SLV couldn't be based on analogy to existing F-15E certified stores
- Aerodynamic data for F-15E/SLV loading needed
 - ♣ Wind tunnel testing not possible, lack of funding
 - ♣ Remaining option was to use CFD to generate aerodynamic data



SLV DIMENSIONS



Weight = 9800 pounds

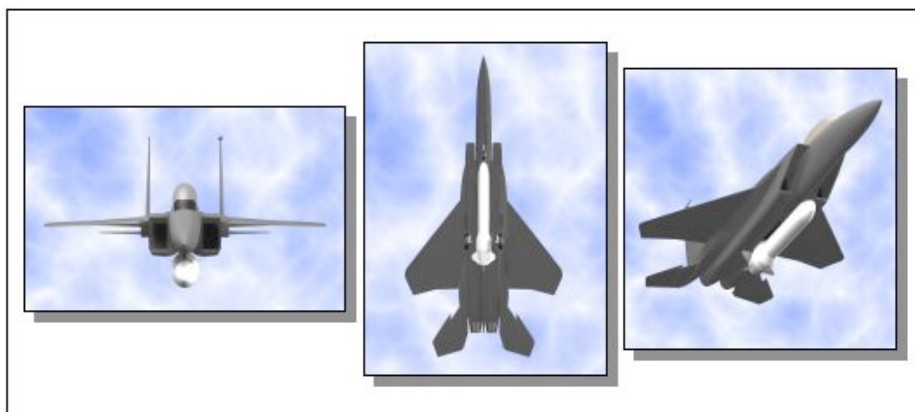


SLV





F-15E/SLV LOADING





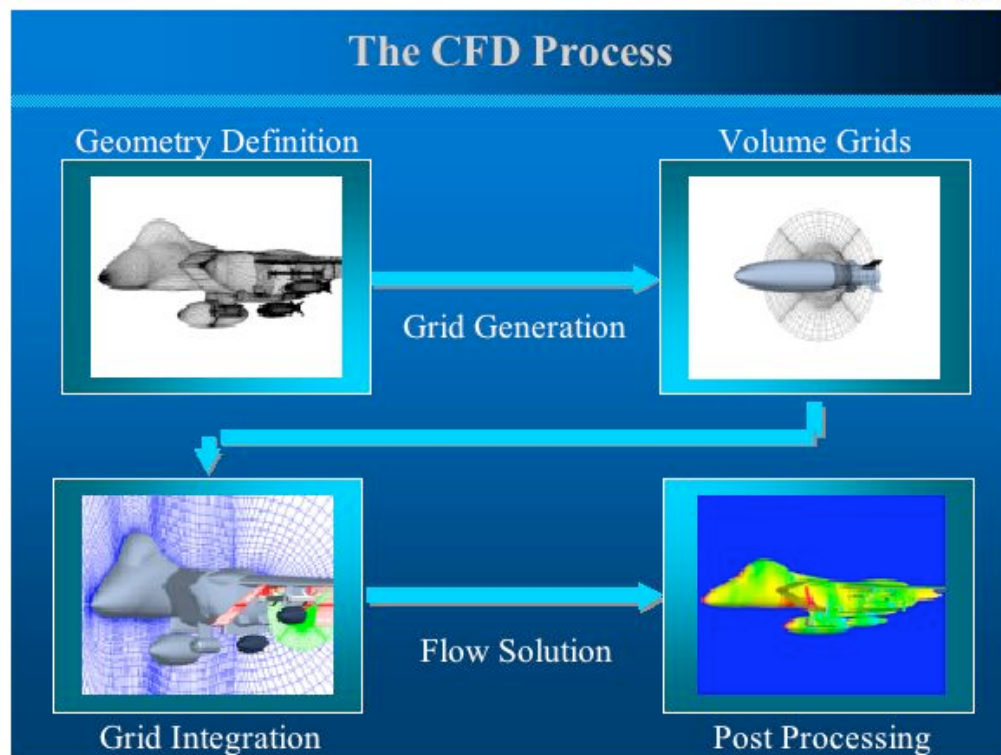
S&C Requirements



- Calculate aerodynamic forces and moments for two loadings
 - ♣ Clean F-15 loading and the F-15+SLV loading
 - ♣ Clean F-15 loading data required to establish baseline CFD data and determine how well CFD data compared with established F-15 aero database
 - ♣ C_x , C_y , C_z , C_m , C_n , C_l
- Use aero data to calculate a SLV store increment
 - ♣ $\Delta SLV = F-15+SLV \text{ data} - \text{clean F-15 data}$
- Aerodynamic data transferred to Boeing/St. Louis to perform flying/handling qualities analysis using six Degree of Freedom (DOF) program



AFSEO CFD Process





The Beggar Code



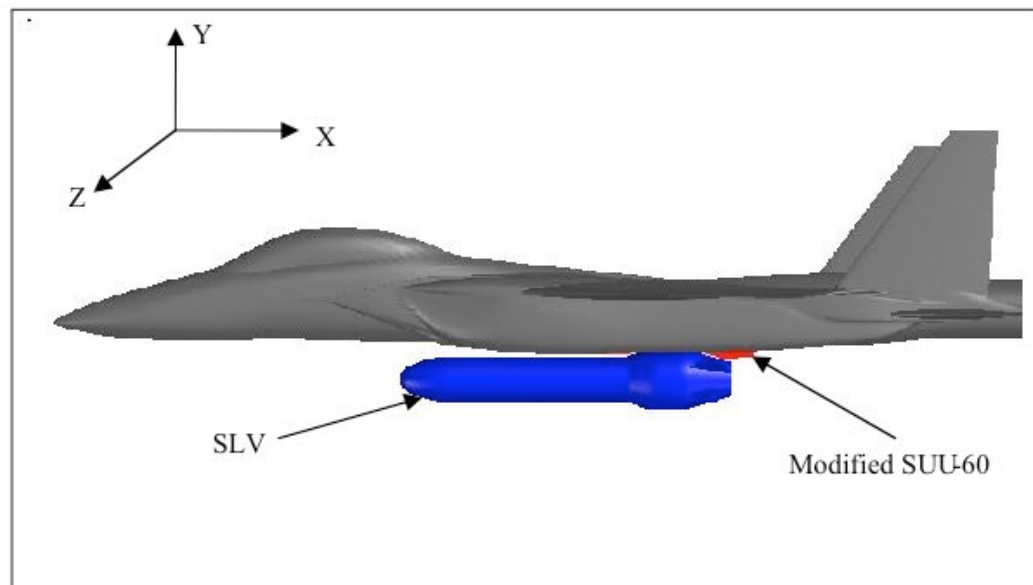
- **RANS, Wall Function or Euler**
 - upwind (2nd order spatial accuracy)
 - Steger-Warming flux vector splitting
 - Symmetric Gauss-Seidel
 - Newton's Method for Time Integration
 - Baldwin-Lomax, Baldwin-Barth, K-Epsilon Turbulence Models
- **Automatic Grid Assembly Scheme**
 - Integrates blocked and embedded "Chimera" grid systems
- **Coupled (6+)DOF with ejector and hinge models**
 - Relative and component motion



Computational Domains



Notice Coord.
System!





Computational Domains



Total System:

37 Grids

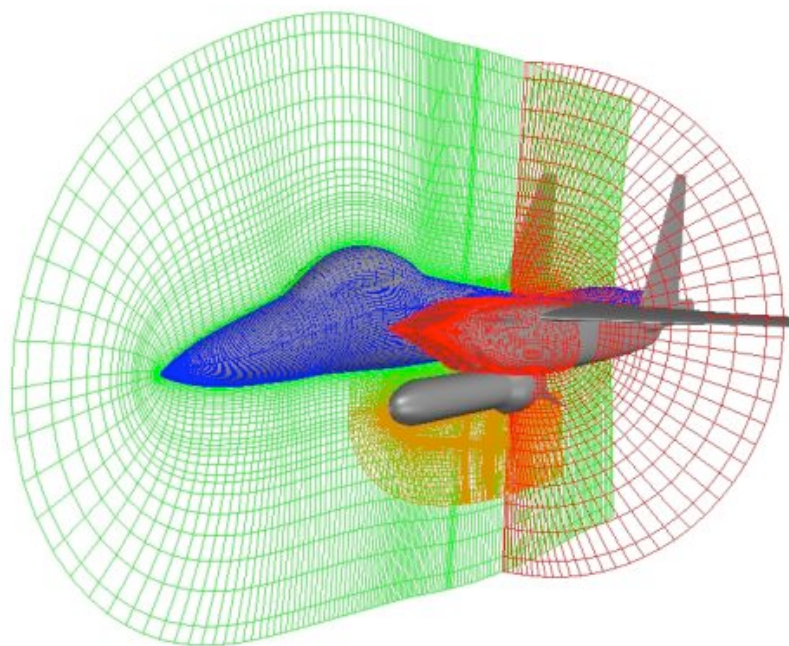
3.1 Mil Points

SLV Alone:

12 Grids

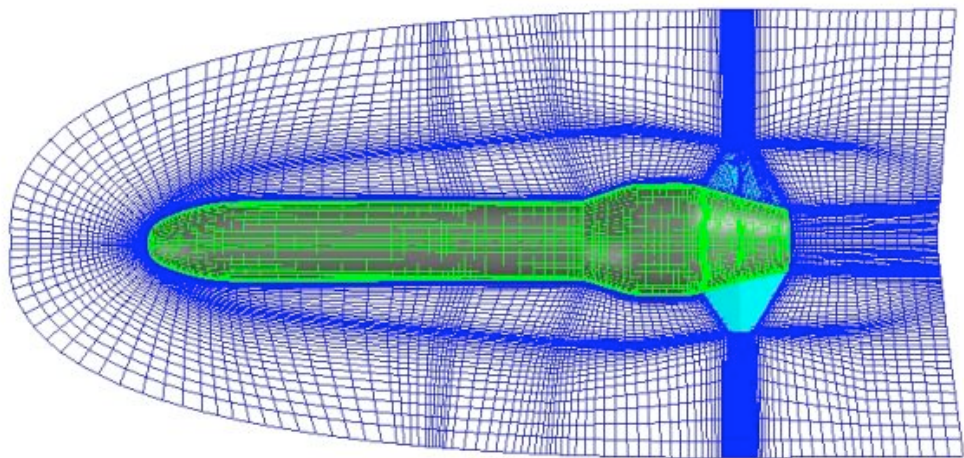
1.1 Mil Points

Inviscid Soln!





Computational Domains





TEST MATRIX



AFSE

Mach	AOA(deg)	AOS(deg)
0.8	-4 to 12	0,4
0.9	-4 to 12	0,4
0.95	-4 to 12	0,4
1.2	-4 to 8	0,4
1.5	-4 to 8	0,4
1.8	0 to 4	0,4
2.0	0 to 4	0,4
2.2	0 to 4	0,4

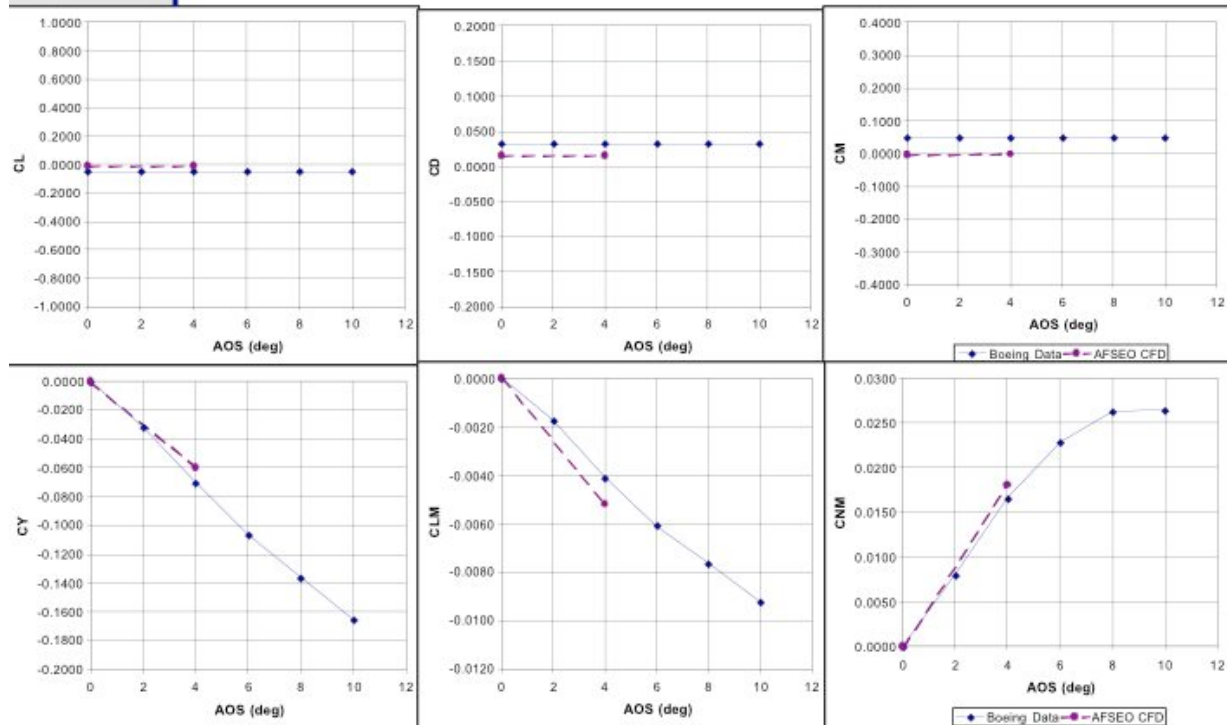
58 Soln's



Comparison With Clean F-15E Aero Database



Mach 0.95 Alpha=0.0

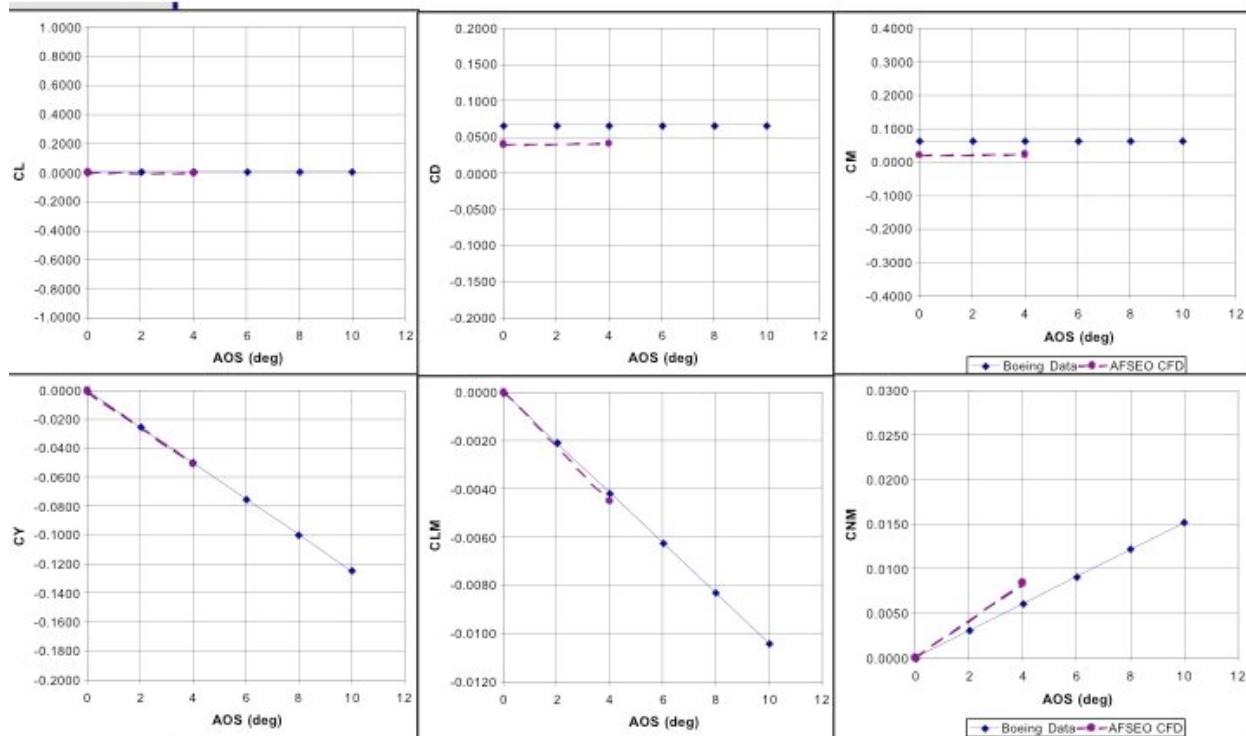




Comparison With Clean F-15E Aero Database



Mach 1.5 Alpha=0.0



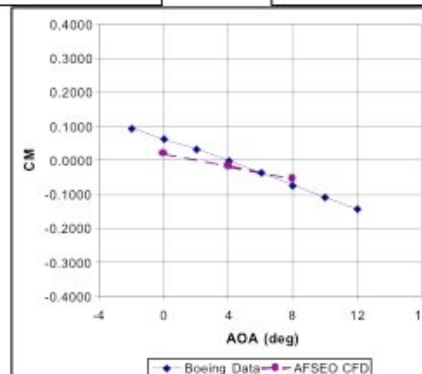
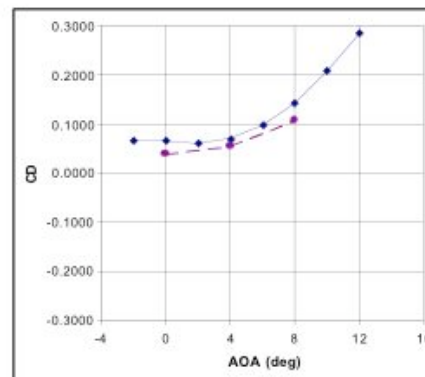
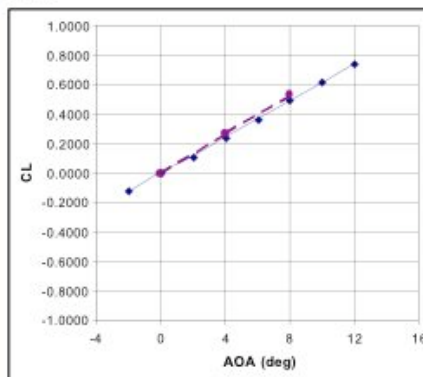


Comparison With Clean F-15E Aero Database



Mach 1.5 Beta=0.0

Lateral
Components
are Zero



Boeing Data AFSEO CFD

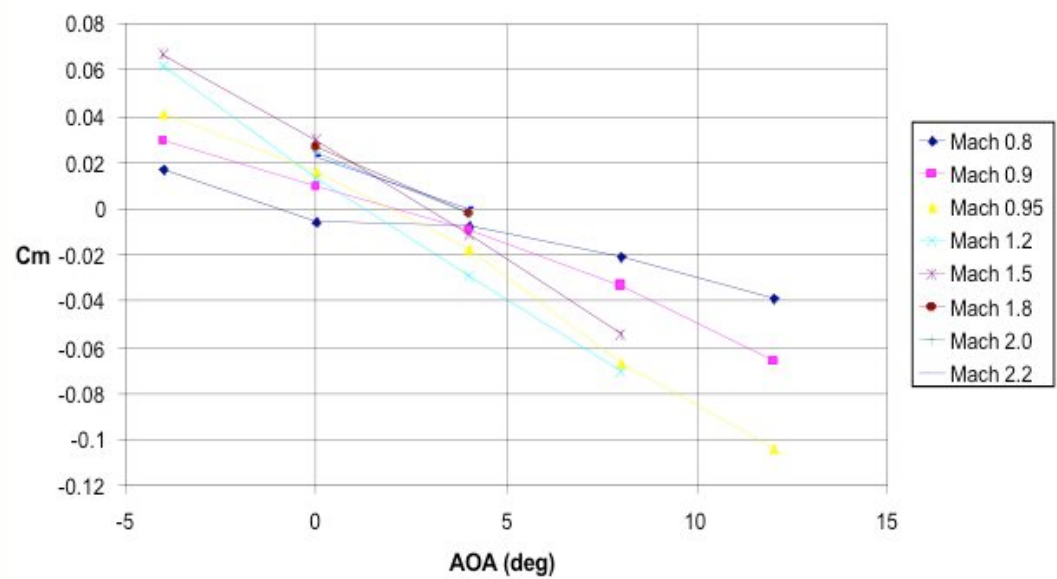


Results



NOTE: Results
are in Stability
Axis Coord.
System

F15E/SLV Loading Pitching Moment Variation



F15E/SLV
loading is
longitudinally
stable



Results

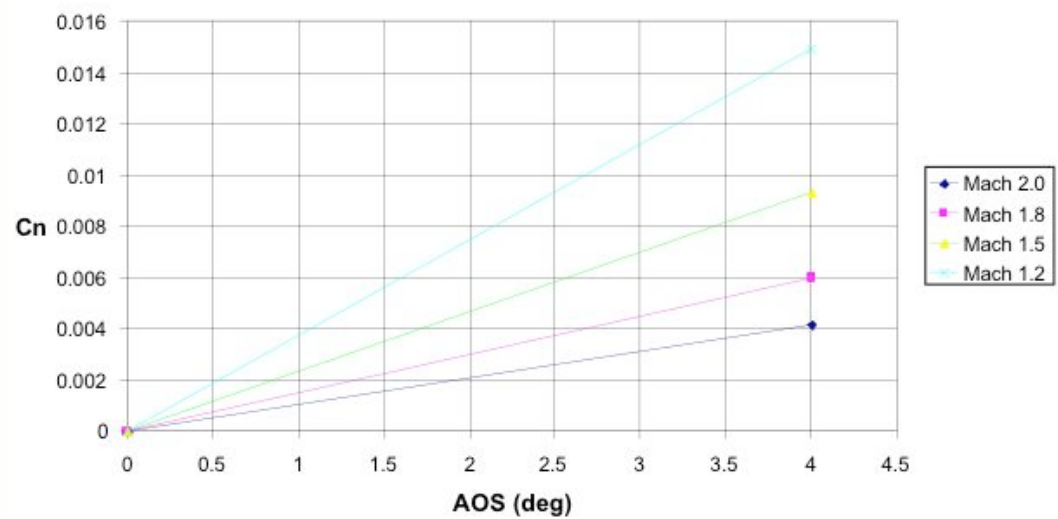


Cn-Beta
Curve

0 deg AOA

F15E/SLV
loading is
directionally
stable but
decreases with
increasing Mach

F15E/SLV Yawing Moment Variation

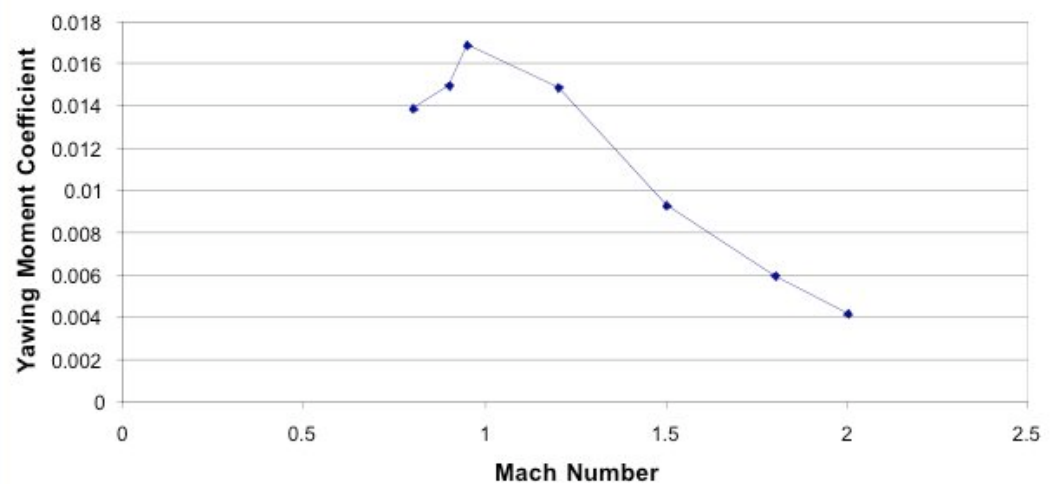




Results



F15E/SLV Yawing Moment Vs. Mach
(AOA=0 Deg and AOS=4 Deg)

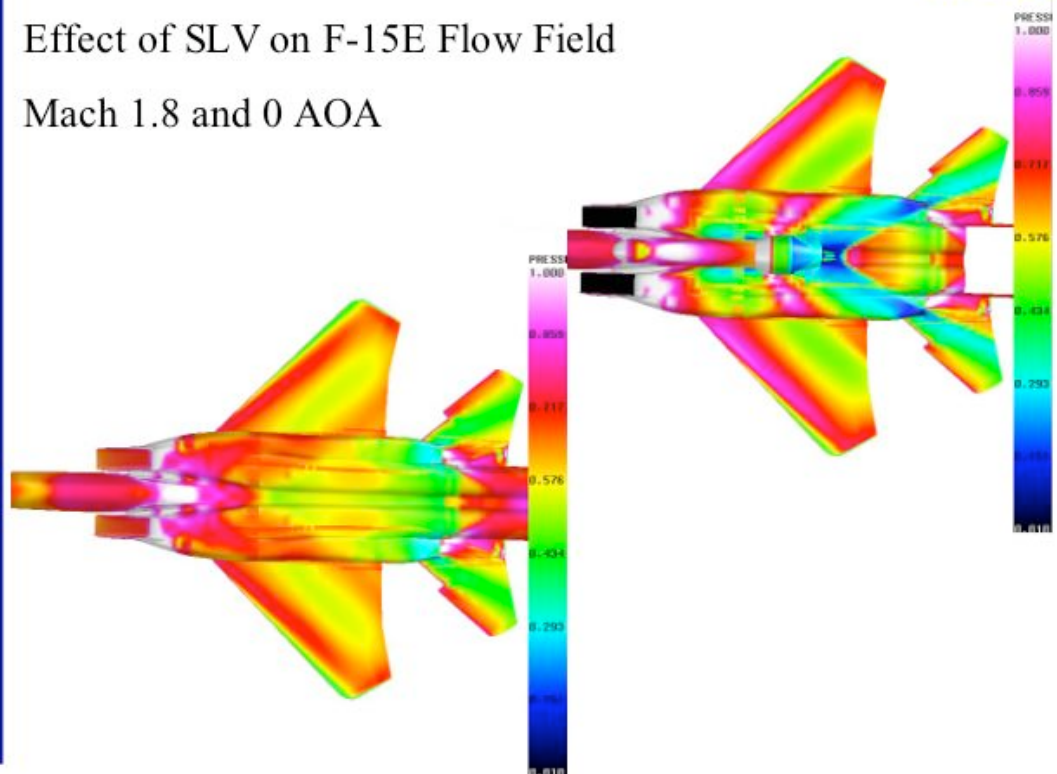




Results



Effect of SLV on F-15E Flow Field
Mach 1.8 and 0 AOA

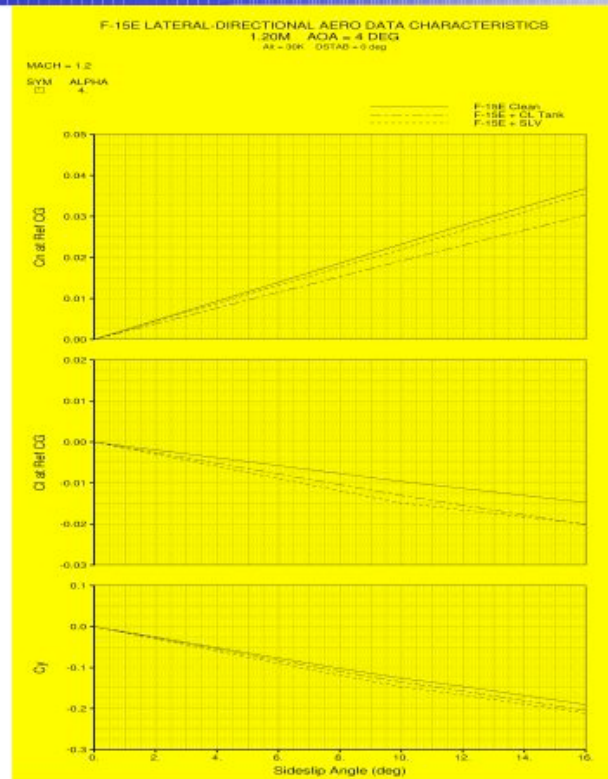




Results



Comparison
of lateral-
directional
coeffs. (Clean
F-15E, F-15E
+ C.L. Tank,
and F-15E +
SLV)

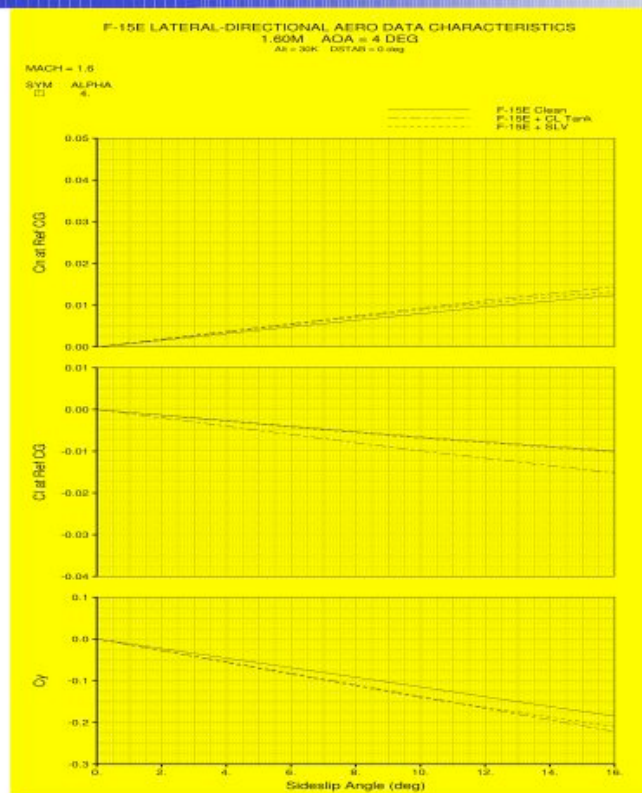




Results



Comparison
of lateral-
directional
coeffs. (Clean
F-15E, F-15E
+ C.L. Tank,
and F-15E +
SLV)





Flying/Handling Quality Results



Boeing/St. Louis Six DOF analysis recommended:

1. Flight allowed to Mach 2.2
2. Limit lateral stick to one-half input above Mach 1.0
3. Angle of Attack (AoA) limited to 30 Cockpit Units

Note: Boeing engineers were surprised how well the baseline F-15E CFD data matched up with the established F-15E aerodynamic database



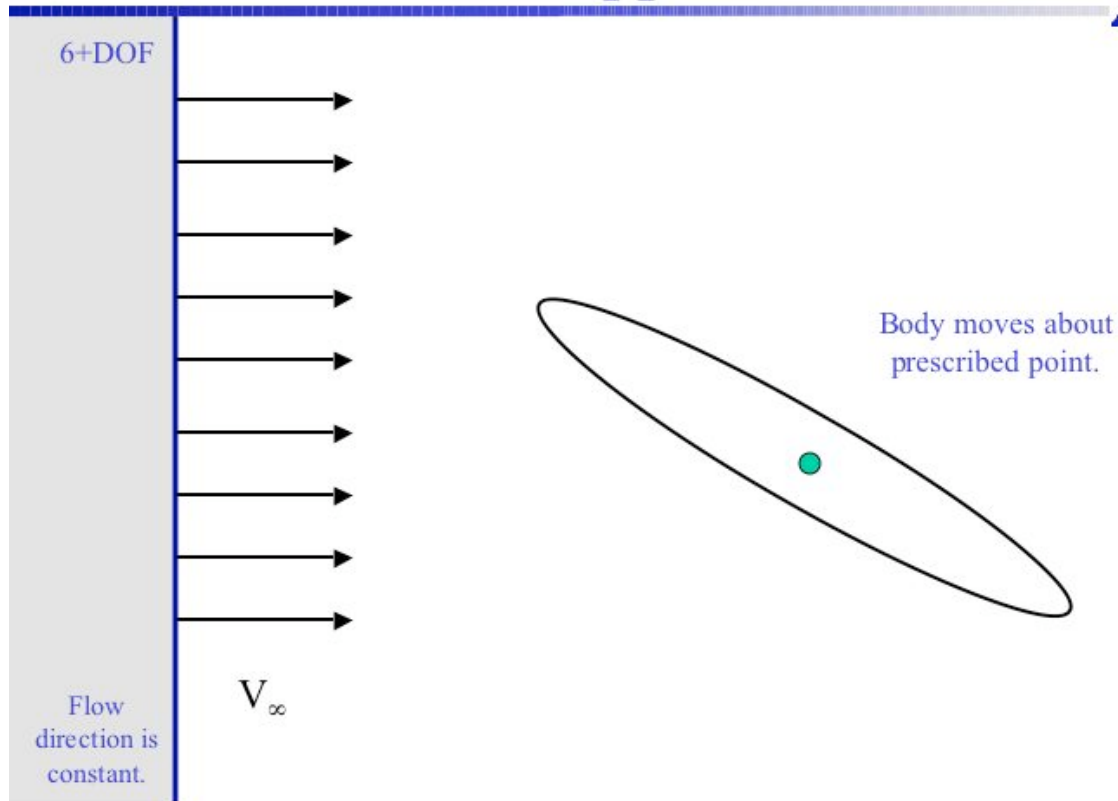
Future Applications of CFD



- Use new 6+DOF to obtain same information quicker. *
- “Delta” analyses for new aircraft/UAVs.
- Commanded control surface deflections. *
- Dynamic stability analysis.
- Maneuvering aircraft.

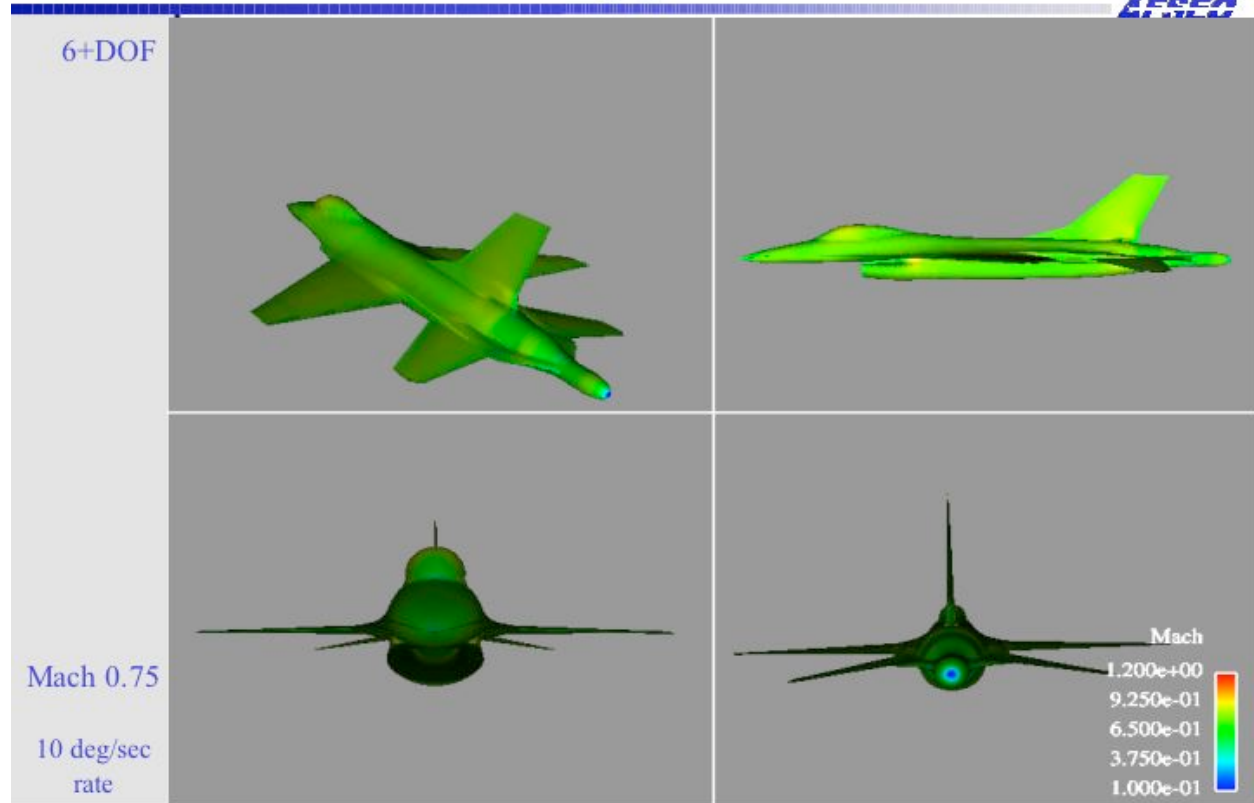


Future Applications





Future Applications



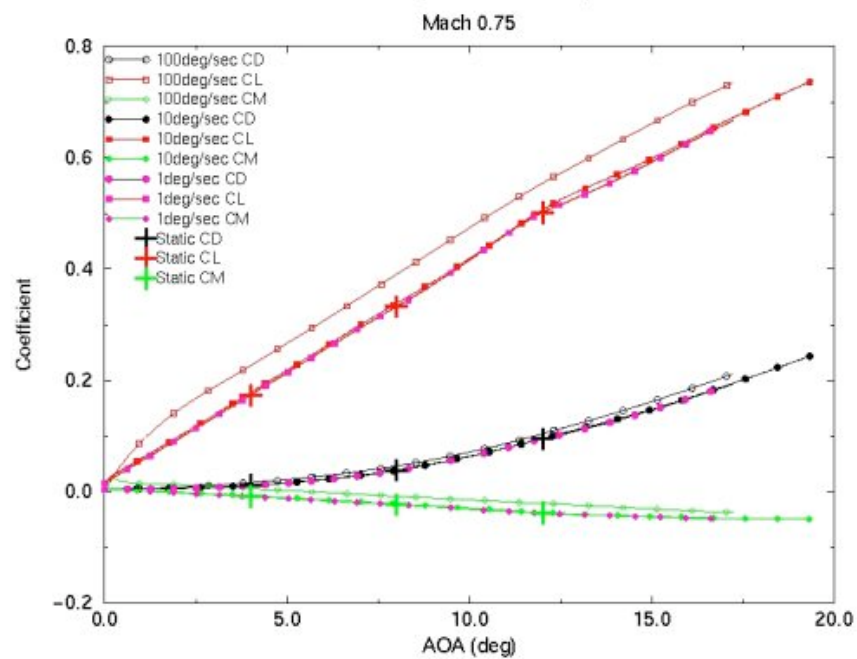


Future Applications



6+DOF

F-16 Moving/Static Comparison

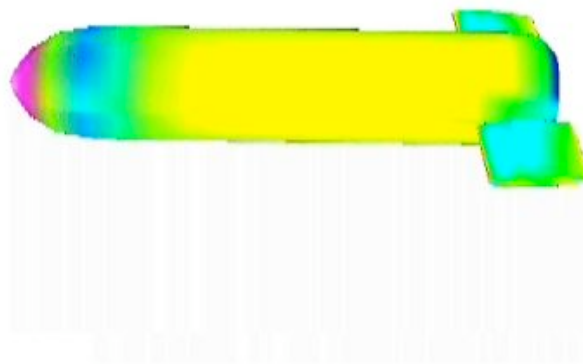




Future Applications



Commanded
deflections



PRES:
0.800
0.663
0.526
0.389
0.252
0.115
-0.022
-0.159

PRES:
0.800
0.663
0.526
0.389
0.252
0.115
-0.022
-0.159

Courtesy of Robert Moran

***Rapid Euler CFD For High-Performance
Aircraft Design***

***Final Report
NASA Langley Research Center
Contract NAS1-96014***

*Eric F. Charlton, Ph.D.
Lockheed Martin Aeronautics Company
Fort Worth, TX
June 29, 1998*

Lockheed Martin Aeronautics Company

Chart Number

Outline



- Motivation
- Configuration
- Resources
- Metrics
- Results
- Falling Leaf
- Discussion
- Future Efforts

Lockheed Martin Aeronautics Company

The goal here was to present one approach to rapid CFD for S&C using an unstructured inviscid method, in order to eventually assess S&C properties as early in the design process as possible. Specific results are presented regarding time, accuracy (as compared to a baseline wind tunnel database) and simplicity for the user. For COMSAC, it's more important to talk about the "specifications" required by Advanced Design and S&C, as well as how the CFD results can be combined for envelope evaluation.

Motivation



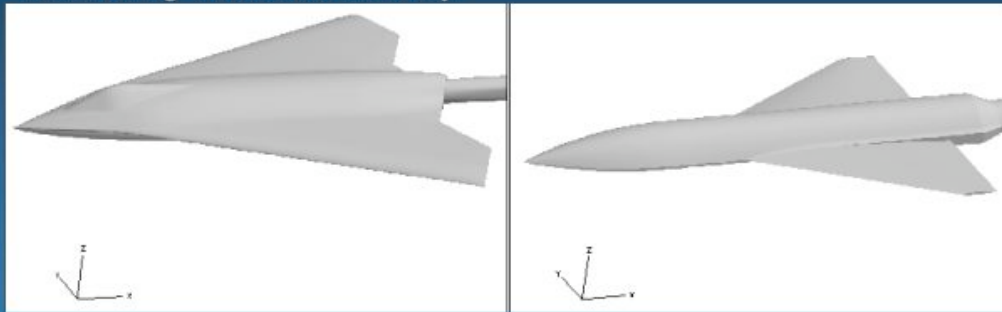
- **Original:**
 - Objective: Rapid Euler CFD To Advance High Performance Aircraft Design.
 - Results: Time, Accuracy, And Ease Of Use.
 - Goal: Routine CFD Usage In Design.
- **Current:**
 - This Project Included Generated Standards Required For CFD To Be Useful To Advanced Design And Stability & Controls.
 - This Project Also Combined CFD Results To Evaluate An Envelope For Stable Flight.

Lockheed Martin Aeronautics Company

Two configurations were considered, the tailless delta wing (“ICE”) configuration and the MTVI configuration. Each configuration actually has two vortices, with the second vortex on the ICE model starting at the change in camber and thickness where the fuselage and the wing blend. Accurate CFD analysis on vortex-dominated flows requires resolving the vortex core; adaptive methods can focus in on the core, but may need to be setup to do so. Vortex breakdown will also be significant in a vortex-dominated flow.

Configurations

- Tailless Delta – “ICE”
 - Blended Fuselage/Cockpit, Broken Trailing Edge.
- MTVI
 - Delta Wing With Chined Body



- Key to Vortex Solution Is Resolving the Core.
 - But How to Adapt to the Core?
- Vortex Breakdown
 - High $\alpha \geq 24^\circ$ and Is Aggravated by Large β (Unsteady).

Lockheed Martin Aeronautics Company

Many CFD cases were run on the two configurations to complete the run matrices.

Run Matrix



MTVI, α -Sweep, Inviscid									
β	α								
0		12		20		30		40	
2	6	12	16	20	25	30	35	40	45

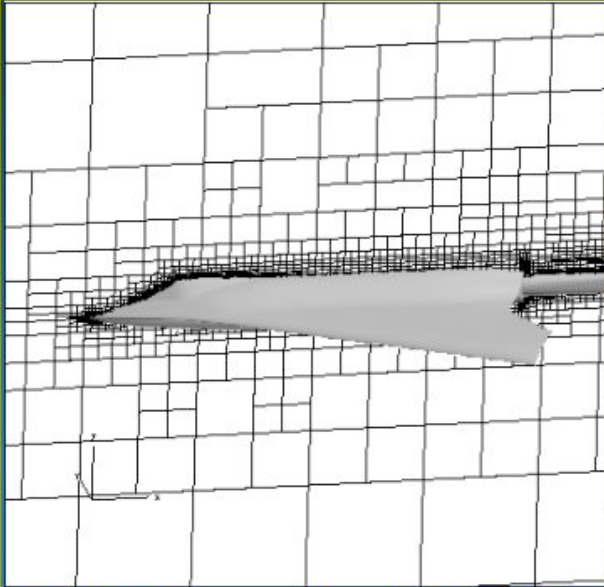
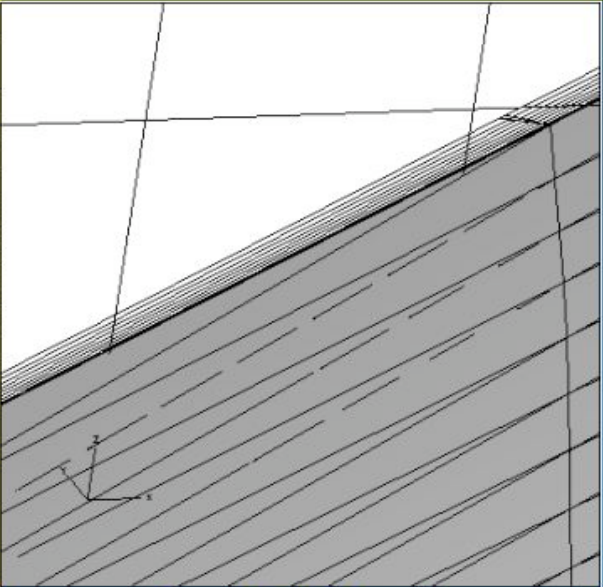
Tailless Delta, α -Sweep, Inviscid									
β	α								
0	0		10	15	20				30
5	0		10	15	20				30
10	0	5	10	15	20	22	24	26	28
20	0	5	10	15	20	22	24	26	28

Tailless Delta, β -Sweep, Inviscid									
α	β								
20	2.5	5.0	7.5	10.0	12.5	15.0	17.5	20.0	

Tailless Delta, Viscous			
β	α		
10	22	24	28
20	22		28

Lockheed Martin Aeronautics Company

Two versions of Splitflow were used; the parallel version does not support the hybrid grids made from prisms extruded from the surface triangulation. The parallel version is still used today on engineering workstations, SGI Origins, and parallel clusters.

Codes and Equipment	
Omnigrid Splitflow Parallel	Hybrid Splitflow Vector
	
SGI, HP/Convex, Cray J-90 (PVM)	Cray J-90 (Vector)

Lockheed Martin Aeronautics Company

The contract effort was divided into two parts, CFD analysis and metrics. The CFD analysis used solution-based adaptation, checked for grid convergence as part of the data comparison, and considered grid resolution as part of the study. Metrics were specified by representatives from two groups, Preliminary Design and Stability & Controls, for the time required, accuracy required, and some measure of the ease of use.

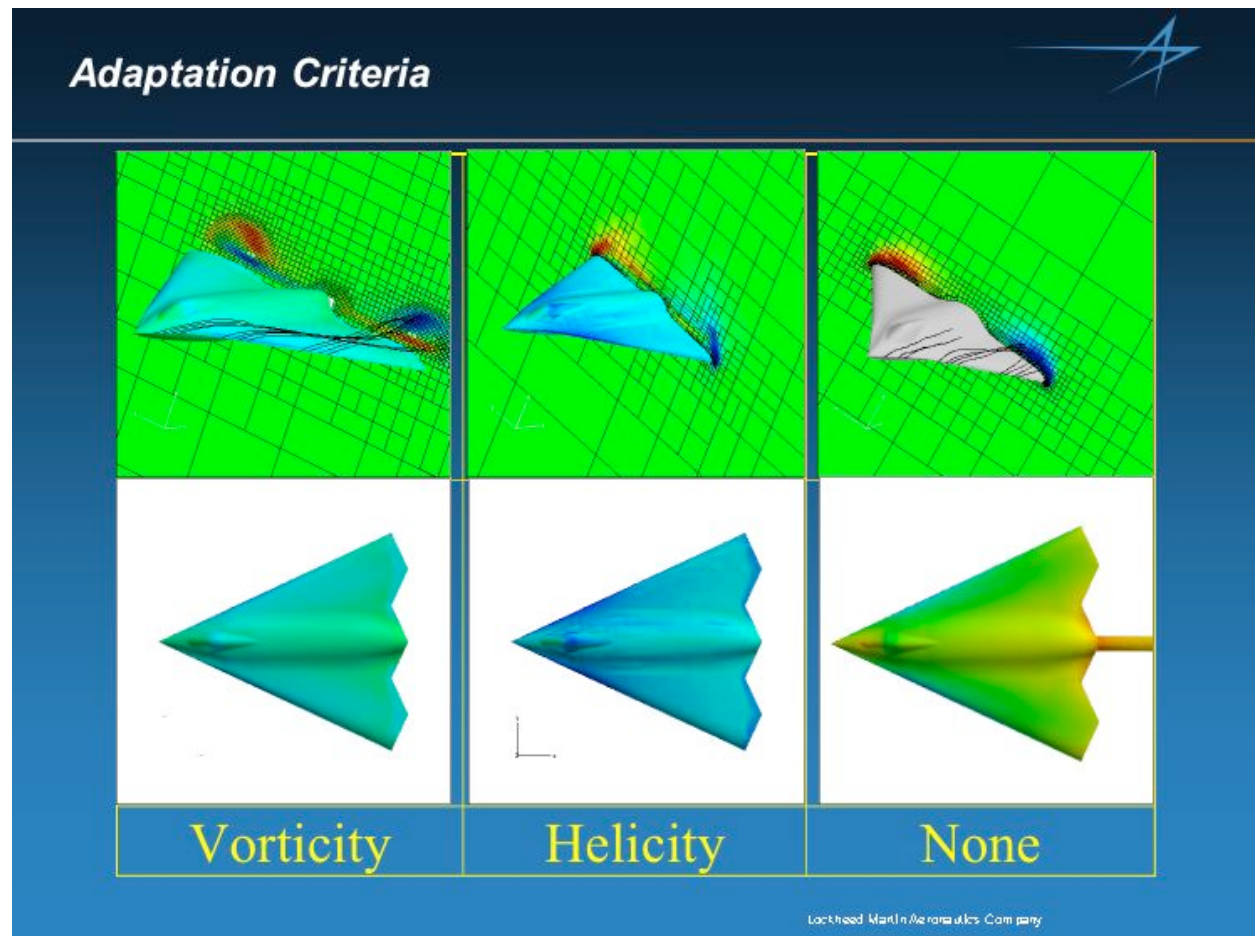
Runtime and Metrics Introduction



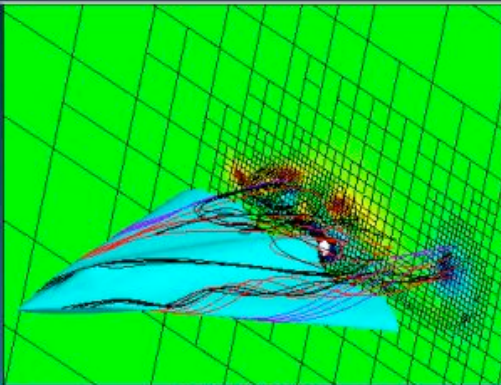
- CFD
 - Solution Adaptation
 - Grid Convergence
 - Leading-Edge Resolution
- Preliminary Design and Stability & Controls
 - Time
 - Accuracy
 - Ease of Use

Lockheed Martin Aeronautics Company

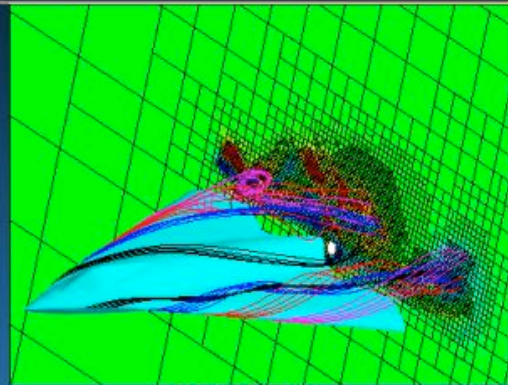
“You cannot solve what you do not resolve.”—Steve Karman. Solution-based grid adaptation gives the grid the opportunity to adapt to the flow solution as the solution progresses. The grid itself helps set what kinds of flow solutions can be modeled, so it is critical to have an appropriate grid. The three plots here show an example of adapting to vorticity, helicity, or not at all, and they show dramatically different results. Traditionally, helicity was used by Splitflow to adapt on vortical structures—the concern was that vorticity would simply add cells to the boundary layer. Unfortunately, helicity does not highlight the vortex core, which is critical for modeling a vortex, and the field value of helicity itself fall to near-noise levels after a vortex burst. Together, those effects make considering raw vorticity critical to adapting to vortex-dominated flowfields.



Tailless Delta wing, Grid Convergence



400K Cells



800K Cells

- 400K Cells
- A Few Cases Grid-converged At 400K And 700K+ Cells.
- No Noticeable Improvement Found In Matching WT Data With The Larger Grid (Any Useful Euler Data Can Be Found At 400K Cells).
- Results Suggest That 400K Cells May Be A Minimum, Due To Necessary Resolution Of The Vortex Core Off The Surface.

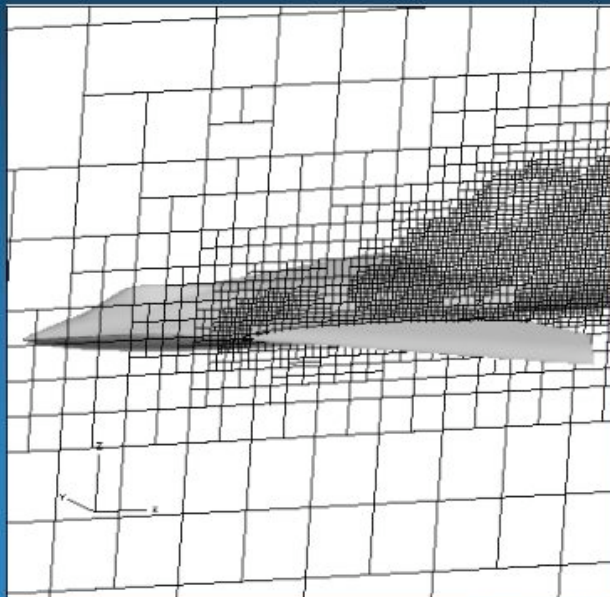
Lockheed Martin Aeronautics Company

Sufficient leading-edge resolution is often critical to setting up the proper flowfield and accurately computing the aerodynamic characteristics on the suction-side of the vehicle. However, the gradients of classic analyzed variables (e.g. pressure, Mach number, density) are not very large compared to those inside of vortices or near shocks. This results in most of the solution-based adaptation going into those areas, not to the leading edge.

Leading Edge Resolution

- Leading-Edge Resolution Is Unimpressive; Adaptation Criteria Favors Adding Cells to the Vortices.

$$M_{\infty} = 0.3, \alpha = 24^{\circ}, \beta = 10^{\circ}$$



Lockheed Martin Aeronautics Company

Metrics are one of the real topics of discussion here. Metrics were developed by consulting with specific colleagues in the areas of Advanced Design and Stability & Control. The metrics presented here could be considered a start for a discussion of more general standards.

Metrics



Metrics Were Required For:

- Time
- Accuracy
- Ease Of Use

With Input From Two Target Groups:

- Preliminary Design
- Stability & Controls

Lockheed Martin Aeronautics Company

Criteria for evaluating CFD for Preliminary Design were produced by discussions with individuals involved in Preliminary Design. At this point, the data presented only represents a few opinions on the matter, but it's a start.

Preliminary Design Criteria (Time)



- Preliminary Design Focuses On The Cruise Point.
 - One Polar Not Enough
 - Depends On Configuration
 - Need Extra Points For High Lift Devices
 - 3 -10 For Flap Schedule And Trim Data
 - 3 Flap × 3 Tail × 3 Leading Edge Flap
 - Possibly Other Ways To Get Tail Data ... Still Leads To A Lot Of Data Required

Lockheed Martin Aeronautics Company

Preliminary Design Criteria (Time) (continued)



- Current Approximate Methods (1998) Require About One Week to Do One Aerodynamic Analysis.
 - The Codes Are Fast
 - Most of the Time Is Spent on Setup, Calibration, and Analysis
- Aerodynamics Wants a Full Matrix in 24 Hours
 - So Need One α Sweep Per Hour

**At This Point, CFD Would Become the Preliminary
Aerodynamic Design Tool**

Lockheed Martin Aeronautics Company

Preliminary design is focused on generating the necessary lift at the minimum drag.

Preliminary Design Criteria (Accuracy)



- Priority For Preliminary Design Is Drag: C_D
 - ΔC_D Might Be Good Enough
- Really Need Lift, Drag, Pitch Moment (C_L , C_D , M_y)
- At A Given C_L , We Want
 - $C_D \pm 5\%$ Below Polar Break (Cruise)
 - $C_D \pm 10\%$ Above Polar Break (Maneuver)
- $C_{D0} \pm 5\%$

Lockheed Martin Aeronautics Company

Preliminary Design Criteria (Ease of Use)



- CFD Tools Often Require Many Training Hours
 - Documentation May Not Be Clear
- CFD Tools Often Have Too Many Parameters To Tweak
 - But Not Enough Parameters To Control the Solution?
- Need Parameter Sets That Work Consistently For A Given Class Of Problem
 - Prefer More Automated Analysis
- A GUI Would Be Nice
 - Perhaps An Expert-Type Input Checker With A Model To Help Set Parameters

Lockheed Martin Aeronautics Company

CFD tools are very useful already, but there are still needed improvements. The user interface is the weak spot, both in getting control information into the code, and in getting configuration information in. These weaknesses affect other codes, too, not just Splitflow, which is very dependent on a “good” surface mesh.

Preliminary Design Criteria (Conclusions)



- Current CFD Tools Produce Useful Answers
 - But Still Need Improvements
- Namelist (Unchecked) Type Input Problems Can Cause Long Delays
 - Formatted Input Is Even Worse
- Need A Way To Reduce Grid Problems
 - Surface Meshing Is Still An Art
 - Automatic Cleanup, Something That Guarantees A Good CFD Ready Surface Mesh
 - Can't Always Refine Out Of A Jam, Need To Fix Problems Up Front
 - Cartesian Methods Like Splitflow Have A Particular Problem With “Split” Cells (Thin Geometry)

Exactly As With Airplane Design in General, We Need a Way to Fix Problems Early in the Process

Lockheed Martin Aeronautics Company

S&C requirements are a little different than Advanced Design, which really focuses on performance. The really tough one is the “within two degrees of zeroes.” When plotted, that band really necks down, and when evaluated, it’s very difficult to accomplish with the methods in this presentation.

S&C Criteria



- S&C Criteria Based on *Proceedings of the Non-Linear Aero Prediction Requirements Workshop*, Edited by Michael J. Logan, NASA Langley Research Center, December, 1993.
- Key Concern Is Reliable Data From a Combination of
 - Databases
 - Linear Methods
 - Higher Order Methods for Potentially Complex Configurations
- Goal Is Simplicity and Speed
 - Antithesis of Euler/NS Methods, Regrettably
- Accuracy:
 - $C_L \pm 5 - 10\%$ Attached, $\pm 10 - 20\%$ Partly Separated, $\pm 20 - 30\%$ Fully Separated
 - Pitch, Roll, Yaw-Moments $\pm 20\%$ of Data Range, Within 2° of Zeroes
 - Doesn't Cover Trim/Performance – Assumed That Once Near, the Control System Can Handle

Lockheed Martin Aeronautics Company

Runtime Metrics Discussion



- Time
 - Setup – Minimal For Splitflow, One Setup Worked For All
 - Run – Much Longer Than Anticipated
 - Rerun – Too Often Required To Compensate For Thin Wing Based Split Cell Problems, BC/Turbulence Models, Etc.
- Accuracy
 - S&C Criteria Included On Plots.
 - Focus Is On Moment Coefficients At $\beta > 0$
 - But Plotted Tighter Tolerance Of Only $\pm 10\%$ To Better Demonstrate Areas Of Desired Improvement
- Ease Of Use
 - Setup
 - Interface
 - Robustness

Lockheed Martin Aeronautics Company

Originally, results were shown for each sweep listed above. Those results are included at the end for completeness but will not be presented here.

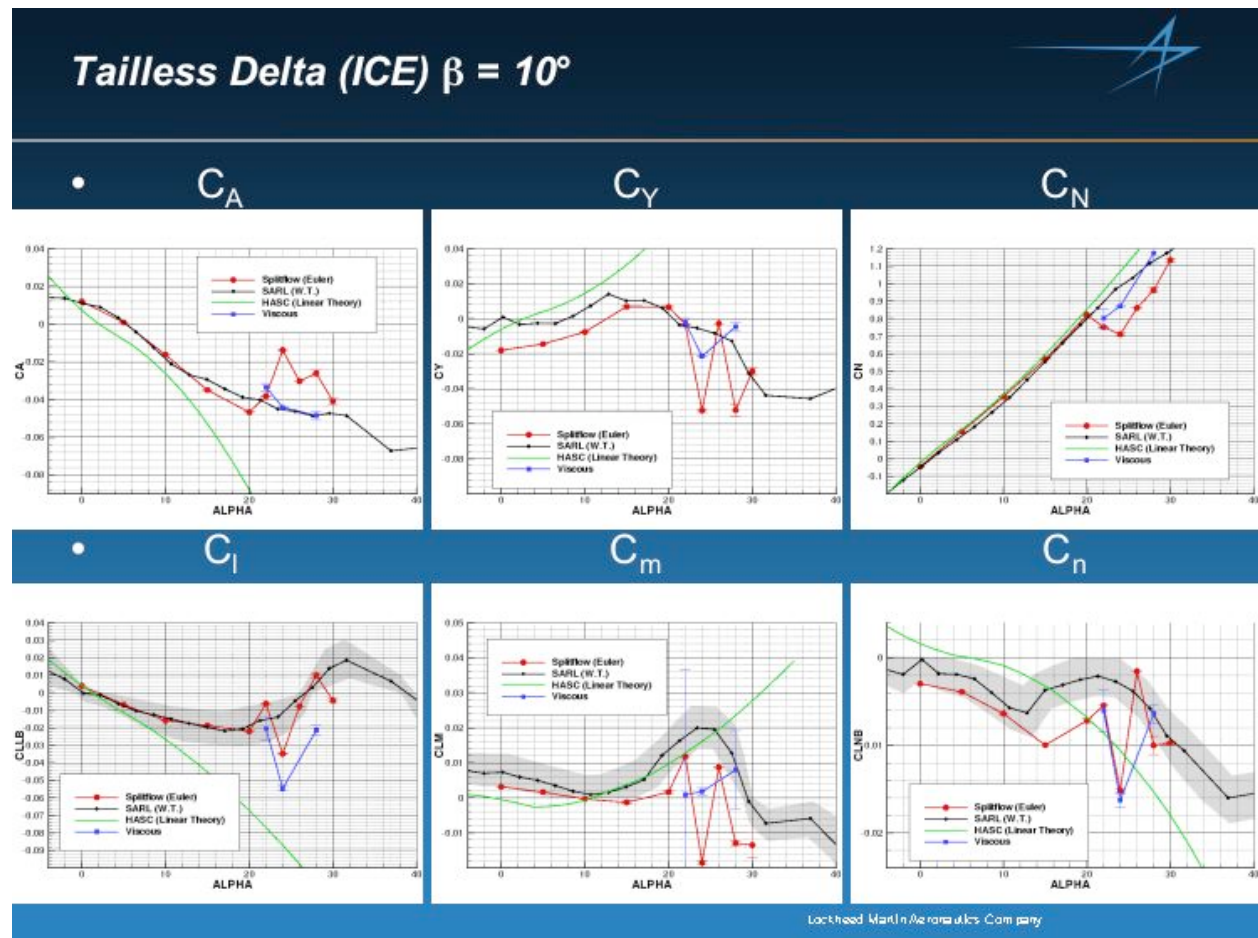
Results



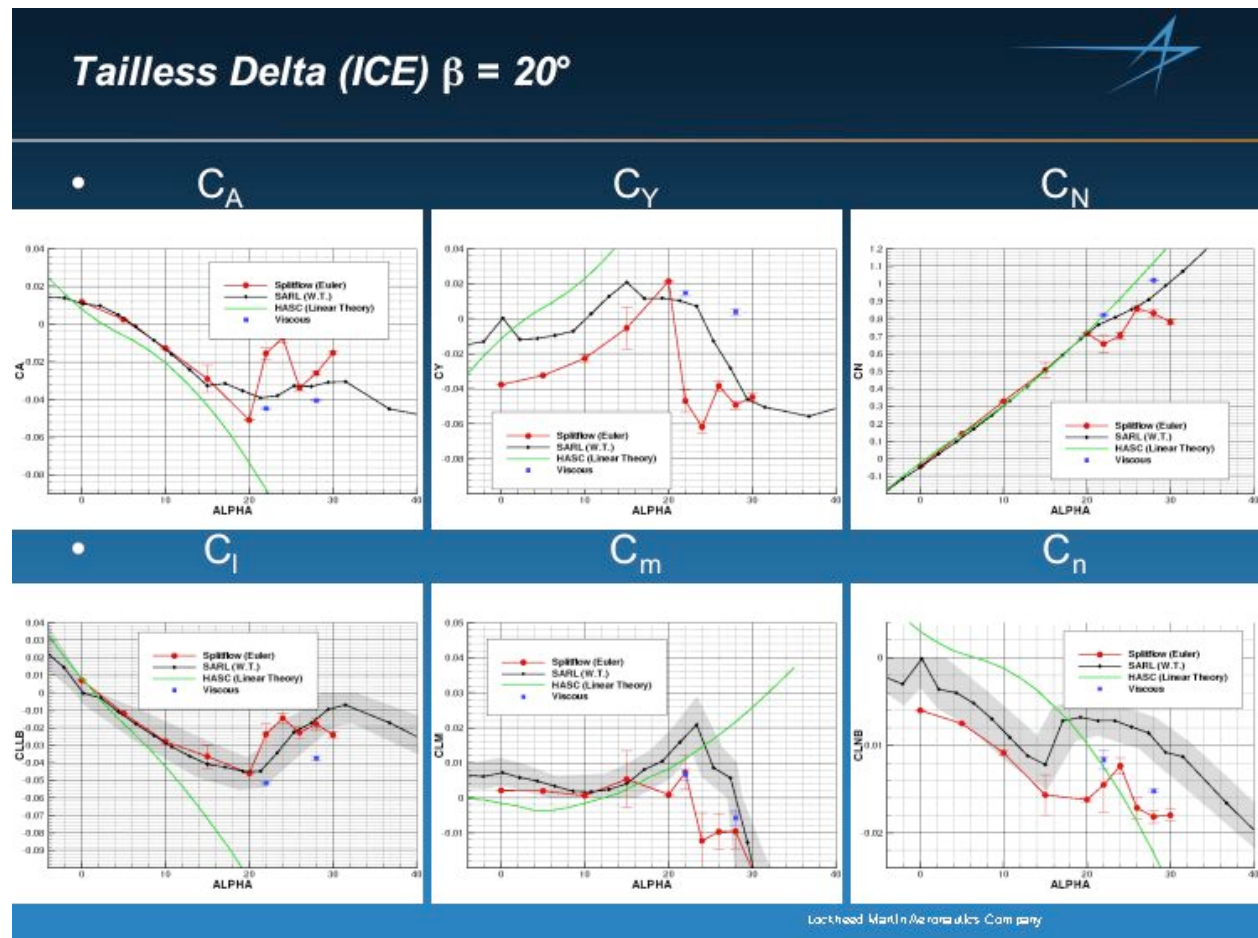
- MTVI, α -Sweep
 - $\beta = 0$
 - $\beta = 2$
- Tailless Delta, α -Sweep
 - $\beta = 0$
 - $\beta = 5$
 - $\beta = 10$
 - $\beta = 20$
- Tailless Delta, β -Sweep

Lockheed Martin Aeronautics Company

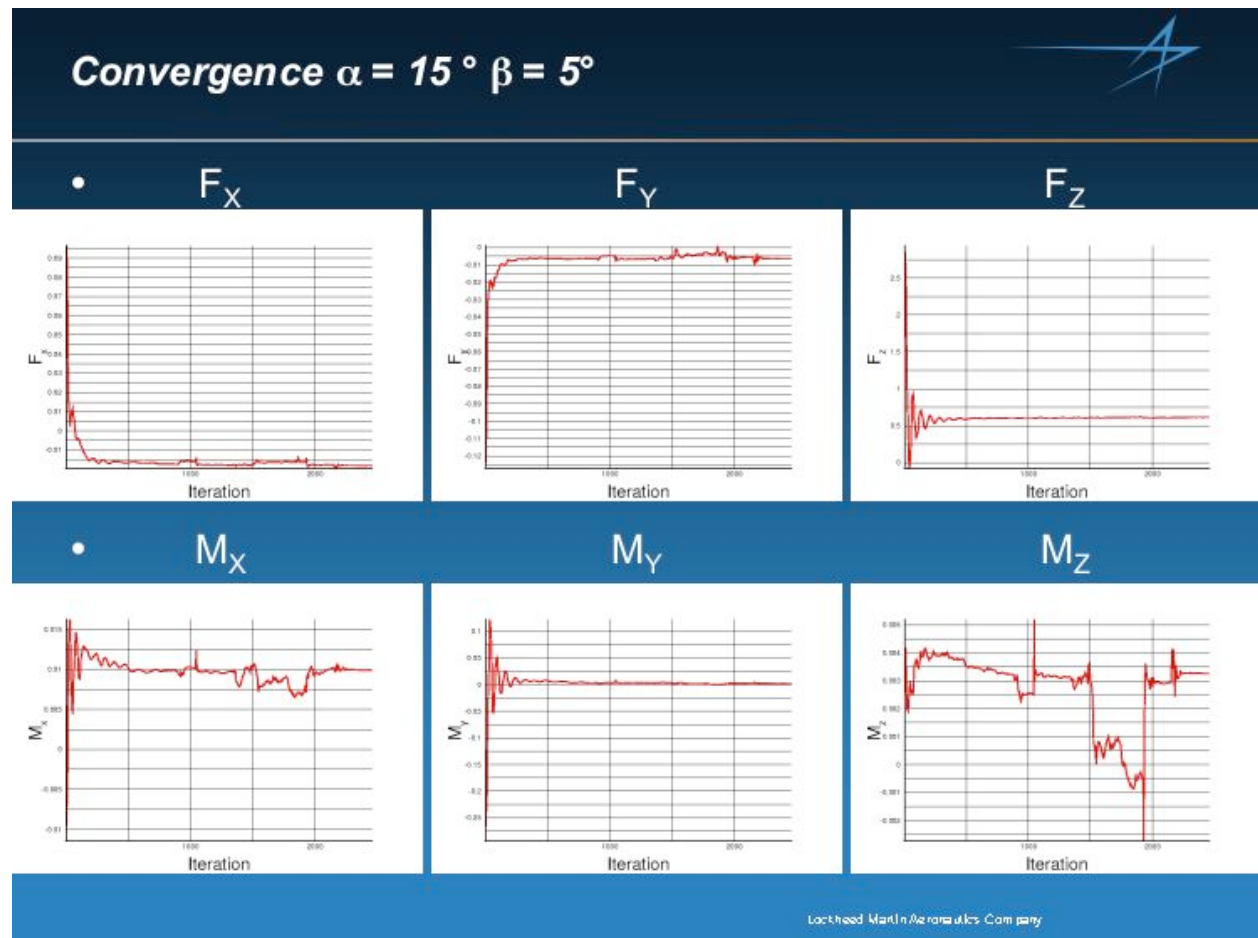
This condition is shown to highlight some of the success and some of the difficulty in this kind of CFD for S&C. Note that the results look “pretty good” until $\alpha > 20$. The error bars indicate how much change there was in the CFD results near the end of the run—note that some of these cases converged very well, just not to something that compares to the wind tunnel data.



This group of results shows just how squirrely some of the CFD analyses were. Note the pitching moment at $\alpha > 10$ in particular, even though the general comparison to the wind tunnel data looks better than it did for the $\beta=10$. Roll Pitch Yaw



This is one example of the kind of convergence experienced on these CFD runs. Note the yawing moment, whose sign is questionable. The value itself is small, however.



Some good results, some bad. At lower AOA, where 24 degrees isn't all that low, the results could be pretty useful. Above 24 degrees, with massive separation, vortex breakdown, and maybe other effects, the results were unpredictable.

Highlighted Results

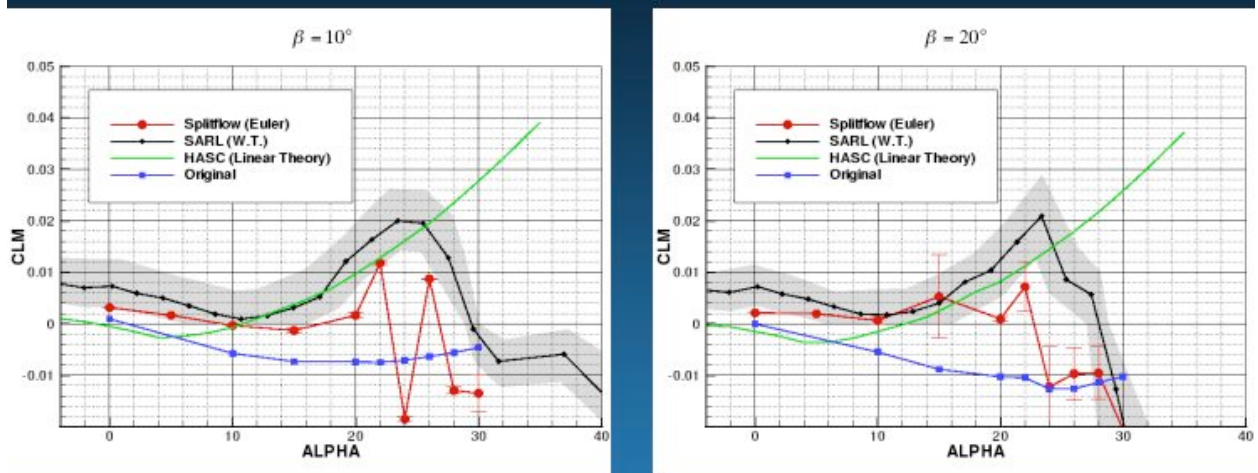


- Which Plots Look Especially Good?
 - Alpha < 24deg, Especially Roll and Yaw Moments
- Which Plots Look Especially Bad?
 - Alpha $\geq$ 24deg, Especially Pitching Moment
- Why?
 - Suspect Vortex Breakdown and Other Unsteady and Viscous Effects. Even at 700K Cells, the Vortex Cores Are Not Well Resolved.

Lockheed Martin Aeronautics Company

This shows some of the emphasis and difficulty in reproducing tunnel results. The error bars on the CFD values show how much the data was changing as the run “converged.” Note that for the left case, all of the runs converged quite well; at 24 degrees, it converged to something completely different from what was measured in the wind tunnel. At 20 degrees sideslip, beyond 14 degrees, the CFD really did not converge very well at all. One thing to keep in mind is that we do not know what the degree of variation of the wind tunnel data was.

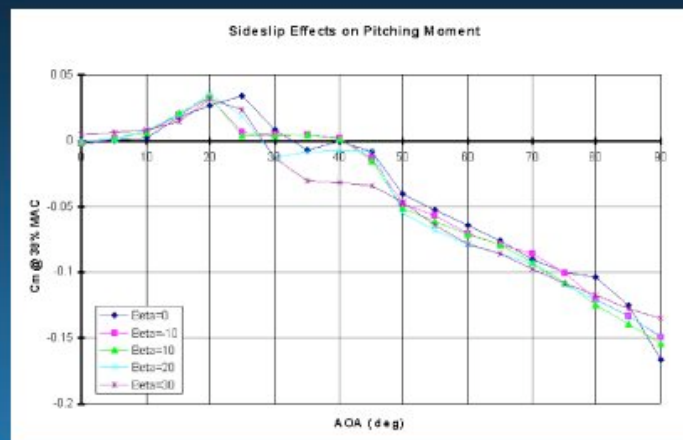
Improved Results



- “Original” Means Quick Run, Coarse (100K Cell) Grid, Four Hour Turnaround on an Octane.

This test of the same ICE model shows that some of the wind tunnel data may not be as certain as it is credited. Note particularly that the 20 and 30 degree sideslip data lie between the 0 and the +/-10 degree sideslip curves, indicating that something happens to change the pitching moment, and then it comes back.

Similar Results

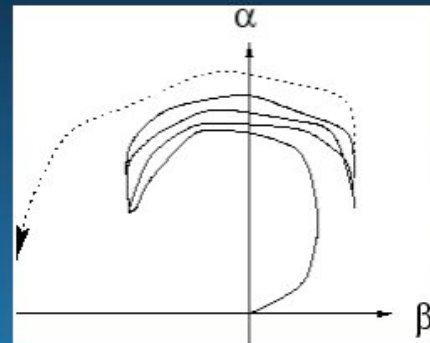
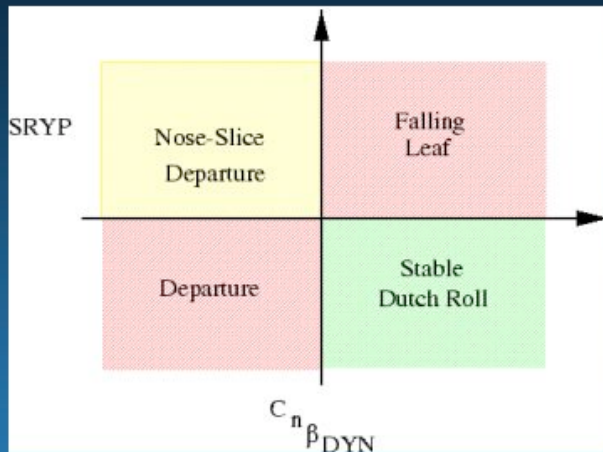


Pitching Moment Data From An ICE Contract Rotary Balance Test, 1996, Test At Bhirle Applied Research 10 Foot Vertical Wind Tunnel In Neuberg, Germany. 1/13th Scale Model, Dynamic And Static Data Collection.

Lockheed Martin Aeronautics Company

Falling Leaf is an extreme flight condition where the airplane rapidly transitions between high beta and low alpha, to high alpha and zero beta, on to high alpha and high negative beta, and then back. The Sustained Roll-Yaw Parameter measures the susceptibility to Falling Leaf, compared to simple departure or a stable Dutch roll.

Falling Leaf



- SRYP = Sustained Roll-Yaw Parameter
- Material from: Foster, John V., "Analysis of 'Falling Leaf' Motion of Class IV Airplanes, Status Update," Flight Dynamics Working Group Meeting, NAWC, Patuxent River, MD, July 27, 1995

Lockheed Martin/Aeronautics Company

These are two definitions and the numeric values for the ICE configuration.

Falling Leaf Parameters



- Roll Moment: $C_{l\beta} = \frac{\partial C_l}{\partial \beta} \simeq \frac{\Delta C_l}{\Delta \beta} \simeq \frac{C_l}{\beta}$
- Yaw Moment: $C_{n\beta} = \frac{\partial C_n}{\partial \beta} \simeq \frac{\Delta C_n}{\Delta \beta} \simeq \frac{C_n}{\beta}$
- Roll Moment Of Inertia (slug*ft^2): $I_{xx} = 35,479$
- Yaw Moment Of Inertia: $I_{zz} = 110,627$
- Cross Product Moment Of Inertia: $I_{xz} = -525$

Lockheed Martin Aeronautics Company

Then, defining those two parameters, we can check for susceptibility to Falling Leaf.

Falling Leaf Criteria



- Define Dutch-roll Stability Parameter:

$$C_{n\beta_{\text{DYN}}} = C_{n\beta} \cos \alpha - \frac{I_z}{I_x} C_{l\beta} \sin \alpha$$

- And Sustained Roll-Yaw Parameter:

$$\text{SRYP} = \frac{\frac{C_{n\beta}}{C_{l\beta}} + \frac{I_{xz}}{I_{xx}}}{\frac{I_{zz}}{I_{xx}} + \frac{I_{xz}}{I_{xx}} \frac{C_{n\beta}}{C_{l\beta}}}$$

- Where Falling-Leaf Susceptibility Is:

$$C_{n\beta_{\text{DYN}}} > 0 \quad \cap \quad \text{SRYP} > 0$$

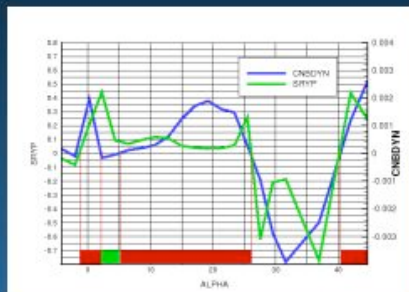
Lockheed Martin Aeronautics Company

Now we can take both the wind tunnel dataset and the Splitflow dataset, compute the Dutch-roll parameter and the “syrup” parameter, plot them together, and determine the envelope susceptible to Falling Leaf. Here, the first important thing to note is that the Splitflow envelope prediction was pretty close to the wind tunnel; the second is that an active control system would be required for all angles of attack for this aircraft. Note that the x-scales are different between Splitflow and SARL data

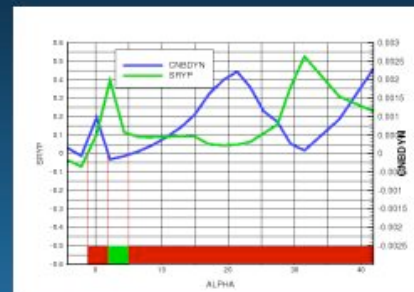
Falling Leaf Results (Tailless Delta)

- Sarl

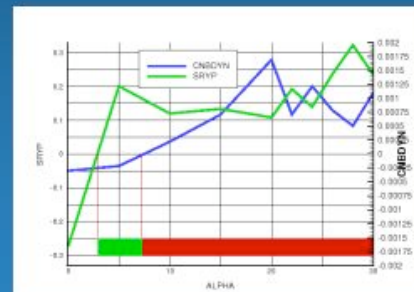
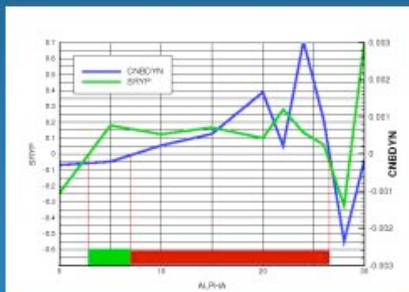
Beta=10



Beta=20



- Splitflow



- Control Augmentation Will Be Required At All Alpha.

Lockheed Martin Aeronautics Company

These methods can produce useful results, and in fact these and similar methods are already in use in the LM Aero Advanced Design and S&C community. It may not be routine yet, and each needs large run matrices that are not yet supported by our CFD tools and computer resources. Another problem is that while this project set out to find a “black box” configuration, no such thing was determined, and in many cases each flight point CFD run required individual attention. Some of the performance applies to five years ago, too. Now, a lot of this analysis can be (and is) performed, opening up these requirements to apply to complete aircraft instead of the simpler configurations (ICE & MTVI) presented herein.

Conclusions



- Much of the Desired Data Can Be Accurately Determined, but Required Larger Grids and Longer Runtime.
- S&C Desires Complete Analysis in Hours – Not Practical With Euler Methods (1998).
- Preliminary Design Wants a Solution Per Hour – Also Not Currently Practical (1998).
- No Black Boxes Here ... Any Minimum Resource Criteria Will Vary With Configuration, Flow Speed, AoA, Etc.
- Goal of Routine CFD Usage in Design Is Coming Along. Now Used by S&C, Advanced Design Has Been Using It for Some Time.

Lockheed Martin Aeronautics Company

Acknowledgements



- Vortex Analysis: Dave Darmofal and Andrew Cary
- Advanced Design: Paul McClure
- S&C: Ken Dorsett
- Surface Preparation: Keith Jordan
- Splitflow Support: Neal Domel, Steve Karman, Brian Smith
- LMTAS Program Manager: Jim Robarge
- NASA Program Manager: Farhad Gaffari

Lockheed Martin Aeronautics Company

The remainder of the presentation consists of support slides.

The MTVI model was run in an alpha sweep at zero sideslip and small (2 degrees) sideslip.

Configuration (MTVI)



MTVI, α -Sweep, Inviscid									
β	α								
0		12		20		30		40	
2	6	12	16	20	25	30	35	40	45

Lockheed Martin Aeronautics Company

The ICE model was run for an alpha sweep at several sideslip angles (0,5,10,20 degrees). A beta-sweep was also done at 20 degrees AOA.

Configuration (Tailless Delta) Inviscid



Tailless Delta, α -Sweep, Inviscid										
β	α									
0	0		10	15	20					30
5	0		10	15	20					30
10	0	5	10	15	20	22	24	26	28	30
20	0	5	10	15	20	22	24	26	28	30

Tailless Delta, β -Sweep, Inviscid									
α	β								
20	2.5	5.0	7.5	10.0	12.5	15.0	17.5	20.0	

Lockheed Martin Aeronautics Company

A few viscous cases were also added to investigate the region near $\text{AOA}=24$ at sideslip, an area where the results compared poorly.

Configuration (Tailless Delta) Viscous



Tailless Delta, Viscous			
β	α		
10	22	24	28
20	22		28

Lockheed Martin Aeronautics Company

Results



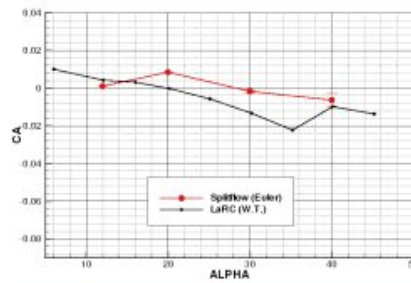
- MTVI, α -Sweep
 - $\beta = 0$
 - $\beta = 2$
- Tailless Delta, α -Sweep
 - $\beta = 0$
 - $\beta = 5$
 - $\beta = 10$
 - $\beta = 20$
- Tailless Delta, β -Sweep

Lockheed Martin Aeronautics Company

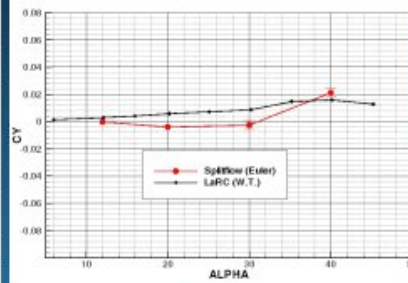
MTVI $\beta = 0^\circ$



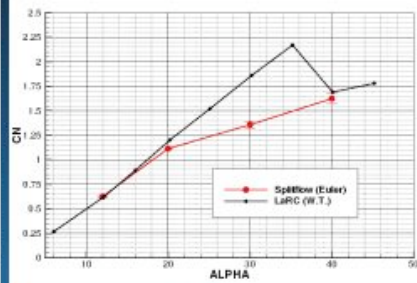
• C_A



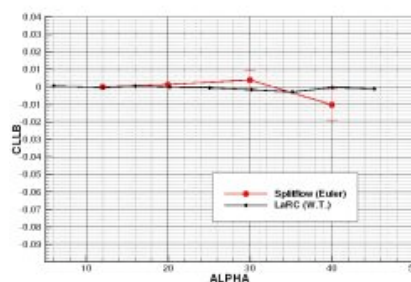
C_Y



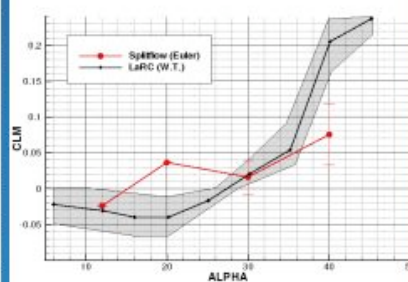
C_N



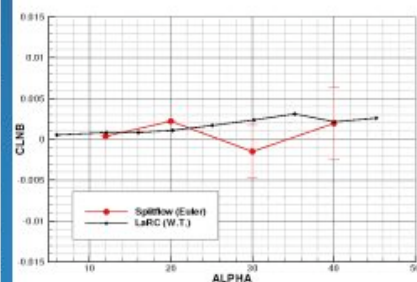
• C_l



C_m



C_n

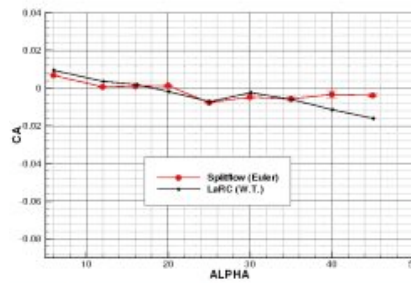


Lockheed Martin Aeronautics Company

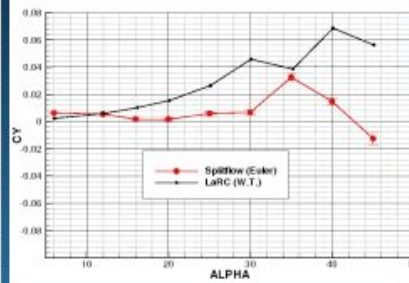
MTVI $\beta = 2^\circ$



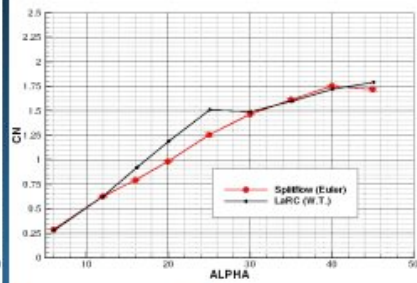
• C_A



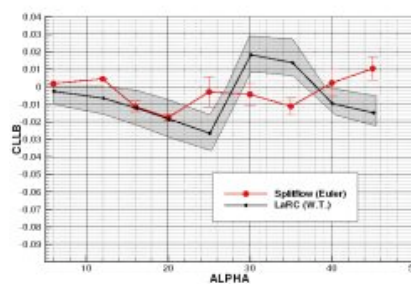
C_Y



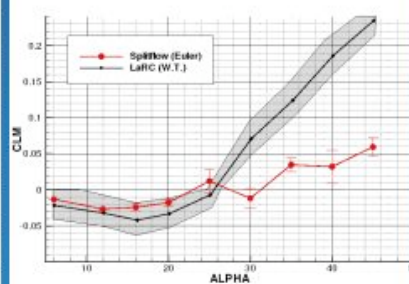
C_N



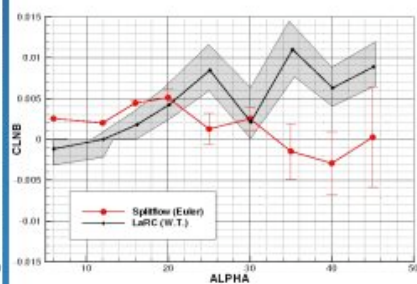
• C_l



C_m



C_n

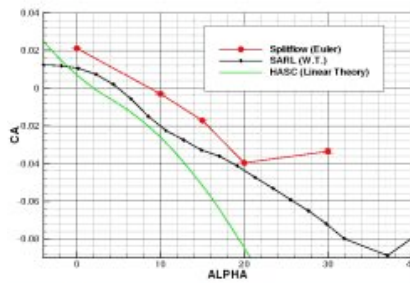


Lockheed Martin Aeronautics Company

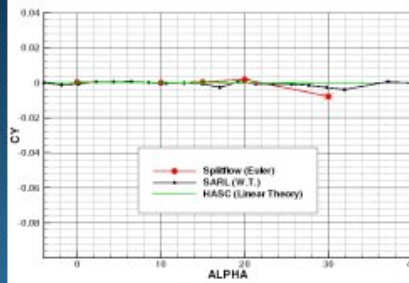
Tailless Delta (ICE) $\beta = 0^\circ$



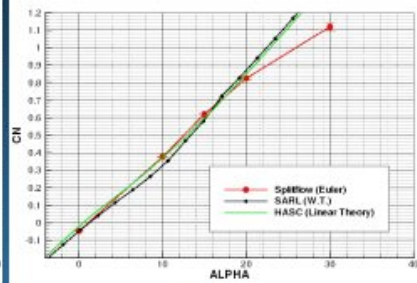
• C_A



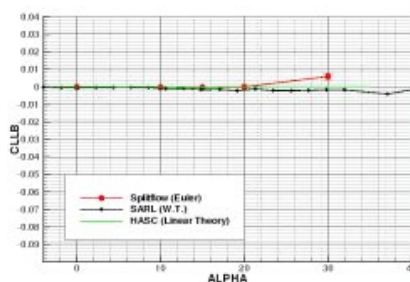
C_Y



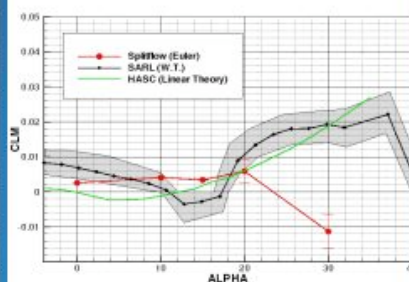
C_N



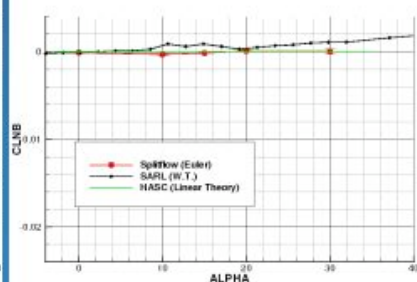
• C_l



C_m



C_n

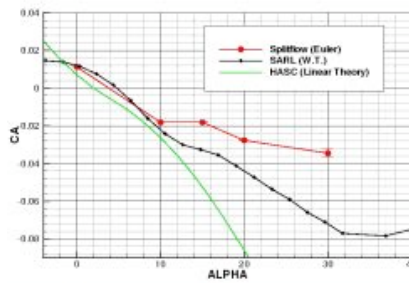


Lockheed Martin Aeronautics Company

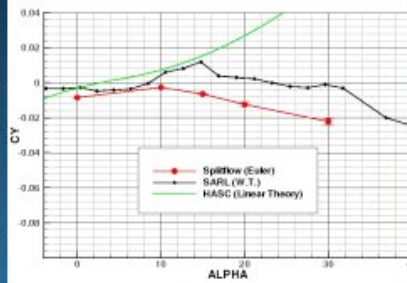
Tailless Delta (ICE) $\beta = 5^\circ$



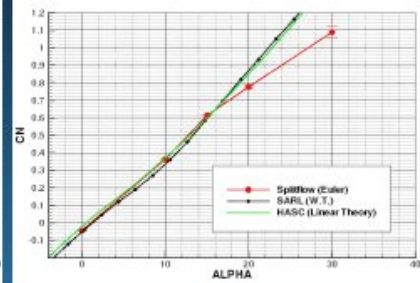
• C_A



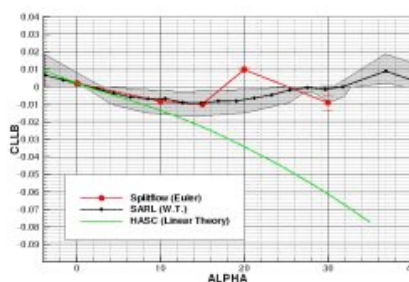
C_Y



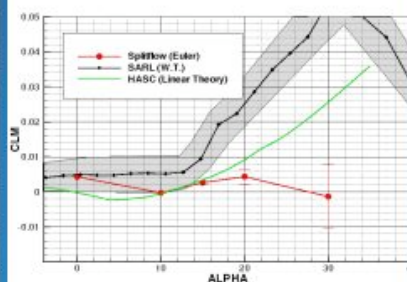
C_N



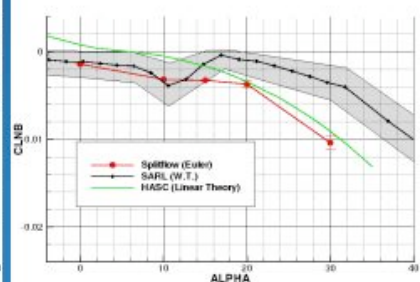
• C_l



C_m



C_n

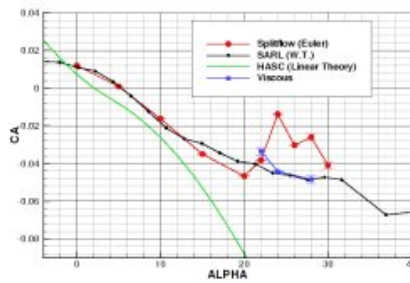


Lockheed Martin Aeronautics Company

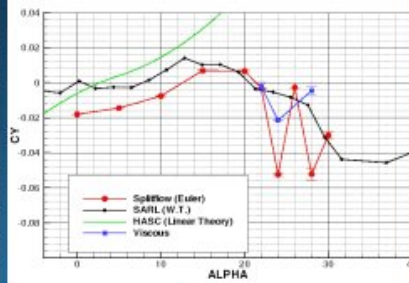
Tailless Delta (ICE) $\beta = 10^\circ$



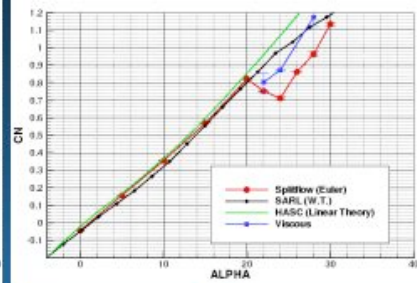
• C_A



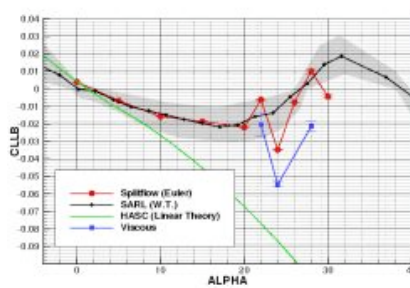
C_Y



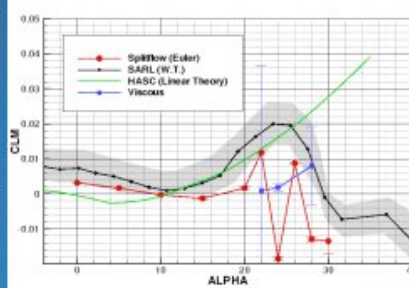
C_N



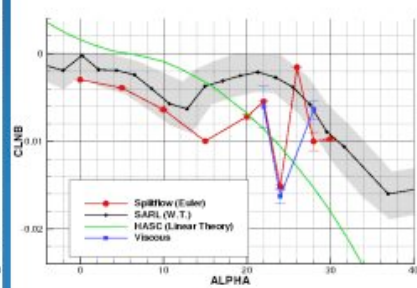
• C_l



C_m



C_n

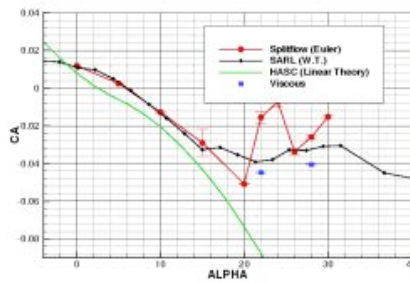


Lockheed Martin Aeronautics Company

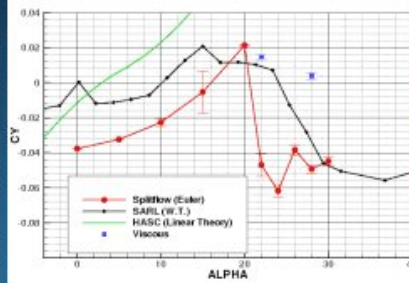
Tailless Delta (ICE) $\beta = 20^\circ$



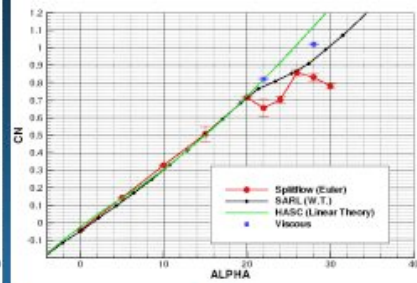
• C_A



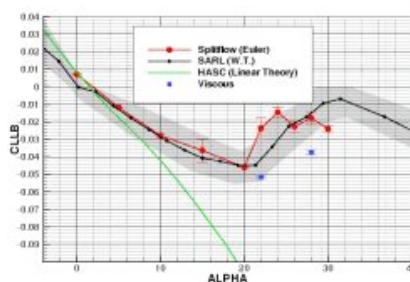
C_Y



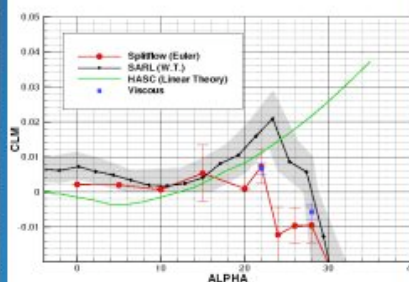
C_N



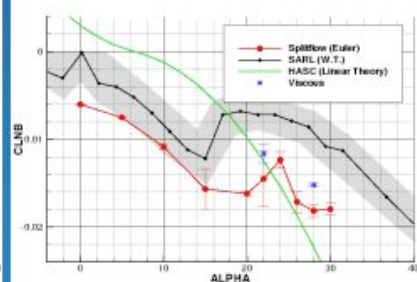
• C_l



C_m



C_n

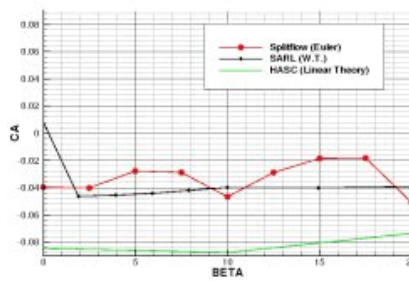


Lockheed Martin Aeronautics Company

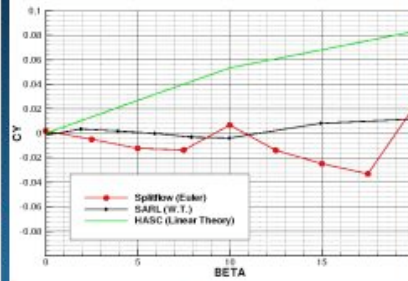
Tailless Delta β -sweep



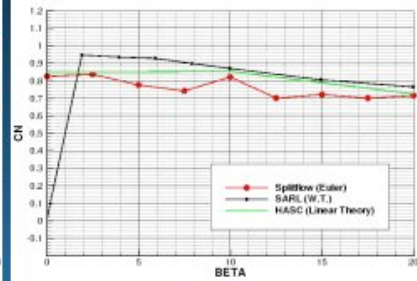
• C_A



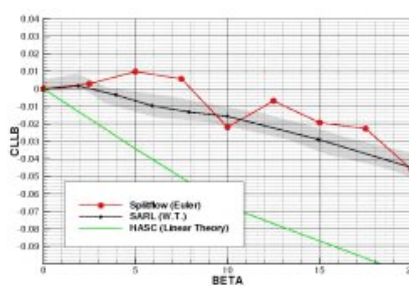
C_Y



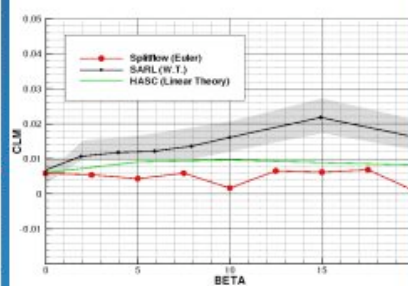
C_N



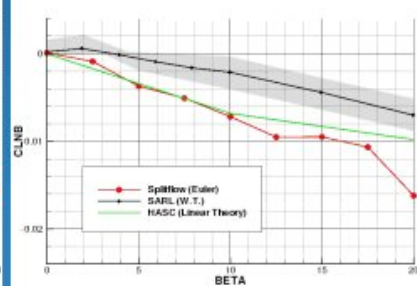
• C_l



C_m



C_n



Lockheed Martin Aeronautics Company

Most of this is old news. Our J90 isn't even used to heat the room anymore. These cases would require about 4 hours on 16 CPU of our Pentium4 cluster, which is very inexpensive.

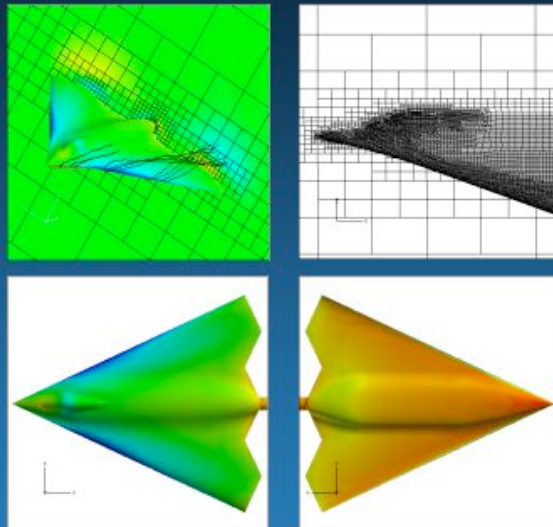
CPU Requirements



- Cray C90 Hours Numbers Were Suspicious.
 - Suspect Trouble in the Accounting System.
- Wall Clock (Real Computing Environment)
 - Approximately 40~130 Hours Runtime in Our Environment
 - (Heavily-Loaded J90, 4~8 Processors on a Parallel Run, Closer to One or Two Weeks Including Queue Waits)
- Affected by
 - Refining Often?
 - Multiple Checkpoint-File Writes, Etc.

Lockheed Martin Aeronautics Company

Future Efforts in Grid Generation



Research work on Splitflow is progressing to build grids that are more acceptable for viscous simulation.

Lockheed Martin Aeronautics Company

5 years ago, this is how this project concluded—we needed faster solutions and more of them. Minimum drag, of course, is now the subject of its own AIAA committee and annual meetings, and it doesn't seem that anyone has a good handle on it yet.

What is really needed?



- Reduced Turnaround Time
 - Smoother Grids?
 - Multigrid?
 - Massively Parallel?
- C_L , C_M
 - Manageable With Proper Technique and Sufficient Patience and Proper Physical Models.
- C_{D0}
 - Statistical Formulation?
 - Laminar NS? RANS? LES? DNS?

Lockheed Martin Aeronautics Company

Our choice of title may seem strange but we mean each word. In this talk, we are not going to be concerned with computations made “after the fact”, i.e. those for which data are available and which are being conducted for explanation and insight.

Here we are interested in preventing S&C design problems by finding them through computation before data are available. For such a computation to have any credibility with those who absorb the risk, it is necessary to quantitatively PREDICT the quality of the computational results.

Quantitative Prediction of Computational Quality (so the S&C folks will accept it)

Michael J. Hemsch, James M. Luckring, Joseph H. Morrison
NASA Langley Research Center



**NASA Symposium on Computational
Methods for Stability and Control
September 23-25, 2003
Hampton, Virginia**

Please note two things:

There are a large number of people at Langley Research Center who are working on these issues, but we got tasked with presenting this talk.

We do not claim that these notions are original to us, but the application and emphasis may be.

We want to make two points here:

1. No answer or a qualitative answer to the question “How good is my answer?” is not good enough for assessing risk. We will address this point in more detail later
2. Where insightful and accurate S&C predictions are most desperately needed is in the design environment. Making a computation after data have been obtained is not a prediction --- it is an explanatory effort. Explanatory efforts can be very useful but they do not require prediction of uncertainty. Note that attempts at calibration do, however, require uncertainty assessments of both the prediction and the experimental data. Otherwise, one has no idea how “fuzzy” the calibration/validation is.



This talk addresses the question:

“How am I going to convince the risk taker that my computation is good enough?”

[for the particular environment of interest.]



Outline

- Risk relates directly to the ability to quantitatively predict uncertainty
- An S&C example and its interpretation based on point-of-view
- Expanding the traditional quality question
- A long-term strategy for creating a controllable process
- A strategy for getting started
 - For getting useful results right now
 - For establishing the long-term process

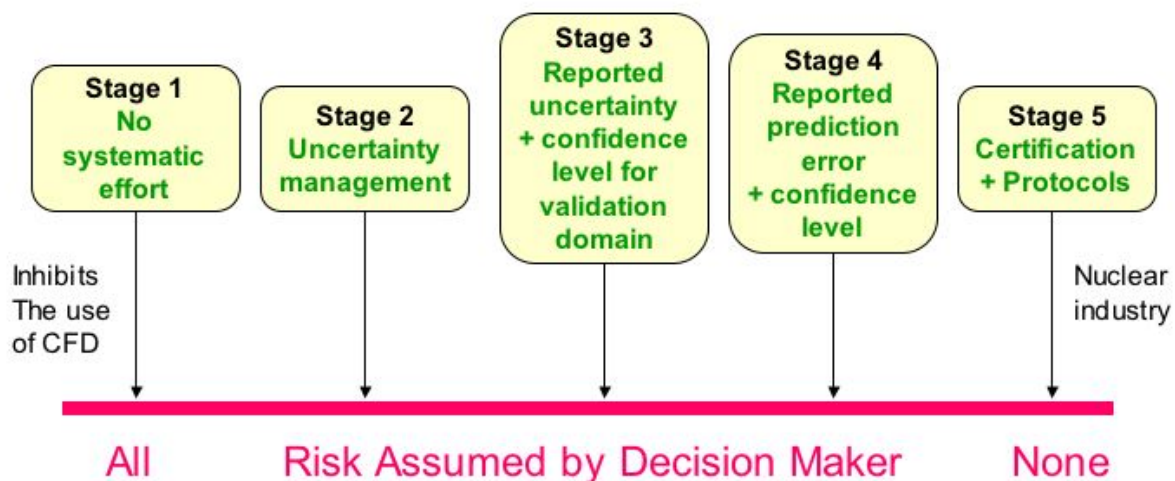
This chart is designed to illustrate the relationship of uncertainty quantification to risk.

At the left end of the chart, there is no defined and managed process in place and no uncertainty quantification is possible. For this state of affairs, the decision maker, i.e. the person or group that uses the computational results, necessarily assumes all of the risk associated with any inaccuracy of the prediction.

At the other end of the chart, the computationalist predicts the uncertainty following protocols and certification procedures suitable for use in a Court of Law. For this state of affairs, the risk is assumed entirely by the computationalist and he or she can be sued.



Why predictive uncertainty quantification (UQ)



The other stages progress from the left to right, but please note that even the very first stage beyond the state of no quantification requires definition of a process and some sort of management system for verifying that the process is being followed.

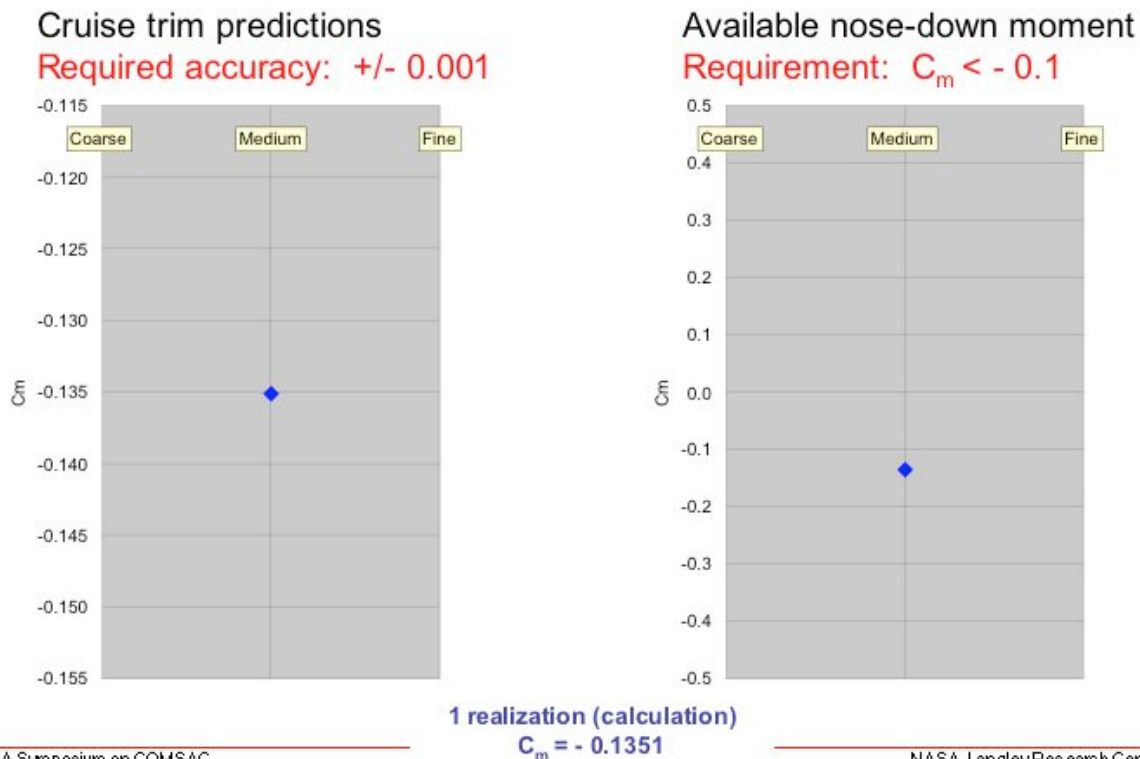
Most (or virtually all) CFD is performed today without quantifying the consequences of uncertainty to an outcome metric. When uncertainty has been considered, it is usually restricted to a limited assessment of grid effects; other sources (turbulence model, algorithm, parameters, user practices, ...) are generally left unaddressed.

The chart contrasts two customer requirements for a CFD computation of pitching moment. On the left is a cruise transport trim application where the required accuracy of the C_m prediction is ± 0.001 . On the right is a high angle of attack S&C application where the performance requirement is to have at least -0.1 nose down authority. The scales are set accordingly for each customer requirement, and the chart also provides for some grid sensitivity information to be added.



CFD without quantified uncertainty

Contrasting Performance and S&C Quality Requirements



In the absence of quantified uncertainty, all that known is the deterministic result that $C_m = -0.1351$. It is not known to any level of confidence if this calculation meets either customer's requirement. Under such circumstances, the prediction is of limited use.

This figure has the identical format to the previous one. However, the results from a fairly extensive uncertainty quantification process are included. Forty-five computations were performed at three different grid density levels. Simple statistics now tell us that $C_m = -0.137 \pm 0.017$ at 95% confidence. This outcome includes a variety of uncertainty sources (different grids, different turbulence models, different flow solvers, etc.)

The individual results are also shown on both the left and right sides of the figure to put them in the context of the two customer requirements. It is clear that the variation is (1) completely unacceptable to the cruise trim requirement on the left and (2) completely acceptable to the high-a S&C objective on the right.



CFD with quantified uncertainty



NASA Symposium on COMS: $C_m = -0.137 \pm 0.017$ (± 2 -sigma coverage) 3A Langley Research Center - 6

Simply put, uncertainty quantification entails determination of conventional terms (average, standard deviation, and confidence) subject to certain process requirements. However, practical techniques will be required to quantify computational uncertainty within available resources and on a timescale consistent to project requirements.

Looking at individual pieces of the quality problem shines more light on them and actually recognizes that they each require different processes and ways of thinking. They end up being separate disciplines which develop on their own and co-evolve as well.



Changing the typical quality question

- **Traditional Question:**
 - How can I get the right answer?
- **New (Totally Separate) Questions:**
 1. How good does my answer need to be?
 2. How do I find out how good my answer really is?
 3. How do I get my answer fast enough?

The question “How good does my answer have to be?” can only be answered by the customer of the computational results, i.e. the risk taker. Of course, the customer should always be informed of the likely quality of the process before he/she commissions the work. Furthermore, the resources provided by the customer can have a significant impact on the possible quality of the computational results.

Unfortunately, the general absence of quantitative predictions of computational uncertainty has led to a typical customer demand of “Do the best you can.” However, recent efforts at several institutions to establish wind tunnel data quality assurance programs have encouraged some customers, most notably performance groups, to revisit their quality needs and to develop well-



Question 1

- **How good does my answer need to be?**
(i.e. “What quantitative quality level is needed?”)
 - The customers, i.e. the user, of the computational predictions are responsible for determining their quantitative quality requirements.
 - Ideally, they would develop a process for doing this on a consistent basis.
 - This has nothing to do with the computational process and is not the subject of this talk.

defined processes for establishing defensible uncertainty requirements. These uncertainty requirements are usually traceable to some design or regulatory requirement that must be met for the airframe program to succeed. Some of these requirements are not even technical in nature, but nevertheless must be met.

Answering this question is the present focus of the computational uncertainty quantification work at Langley Research Center. It is impossible in this short talk to address anything more than the general notions. We recommend the following references:

- P. J. Roache, “Verification and Validation in Computational Science and Engineering”, Hermosa, 1998.
- W. L. Oberkampf, T. G. Trucano, “Verification and Validation in Computational Fluid Dynamics”, Progress in Aerospace Sciences, Vol. 38, No. 3, 2002, pp. 209-272.
- J. M. Luckring, M. J. Hemsch, J. H. Morrison, “Uncertainty in Computational Aerodynamics”, AIAA-2003-0409, January 2003.



Question 2

- **How do I find out how good my answer really is?**

(i.e. “How do I find out the actual quantitative quality level of my prediction?”)

- We need to create a process that can be controlled and evaluated.
- Which process should we use? Do we have to invent a new one?
- What can we do right now to get started?
- This is the main, albeit short, subject of this talk.

- M. R. Mendenhall, R. Childs, “Best Practices for Reduction of Uncertainty in CFD Results”, AIAA-2003-0411, January 2003.
- M. J. Hemsch, “Statistical Analysis of CFD Solutions from the Drag Prediction Workshop”, AIAA-2002-0842.
- M. J. Hemsch, “Statistical Analysis of CFD Solutions from 2nd Drag Prediction Workshop”, AIAA-2004-0556, January 2004.

This question really addresses the issue of what process do I need?

It is possible to use lower-order-physics codes for S&C problems as long as the domain of uncertainty predictability is known in advance. This means that the problem of interest would have to be pretty close to a previously-quantified domain.

For true prediction, when such a previously-quantified domain does not exist, quantified uncertainty prediction does not seem possible.

Note that this notion is especially important when novel configurations are being considered.



Question 3

- **How do I get my answer fast enough?**
(i.e. "How do I get my predictive answer and its uncertainty fast enough?")
 - This is not the subject of this talk, but we can't resist a few comments.
 - Gotta have sufficient physics (or somehow put limit switches on reduced physics codes)
 - Gotta get designers to use them

We recommend the following reference for further reading on process quality assurance:

M. C. Paulk, et al, "Capability Maturity Model for Software, Version 1.1", Technical Report CMU/SEI-93-TR-024 (also ESC-TR-93-177), Software Engineering Institute, Carnegie Mellon University, February 1993 (download from SEI website).

The above referenced document shows how to create and manage such a process. Paulk, et al have applied the approach to the software development process, but it applies just as well to any process, including uncertainty prediction/quantification.



The strategy for the long-term is ---

- Create a process that can be
 - Controlled
 - Evaluated
 - Improved

(i.e. create a **predictable** process)

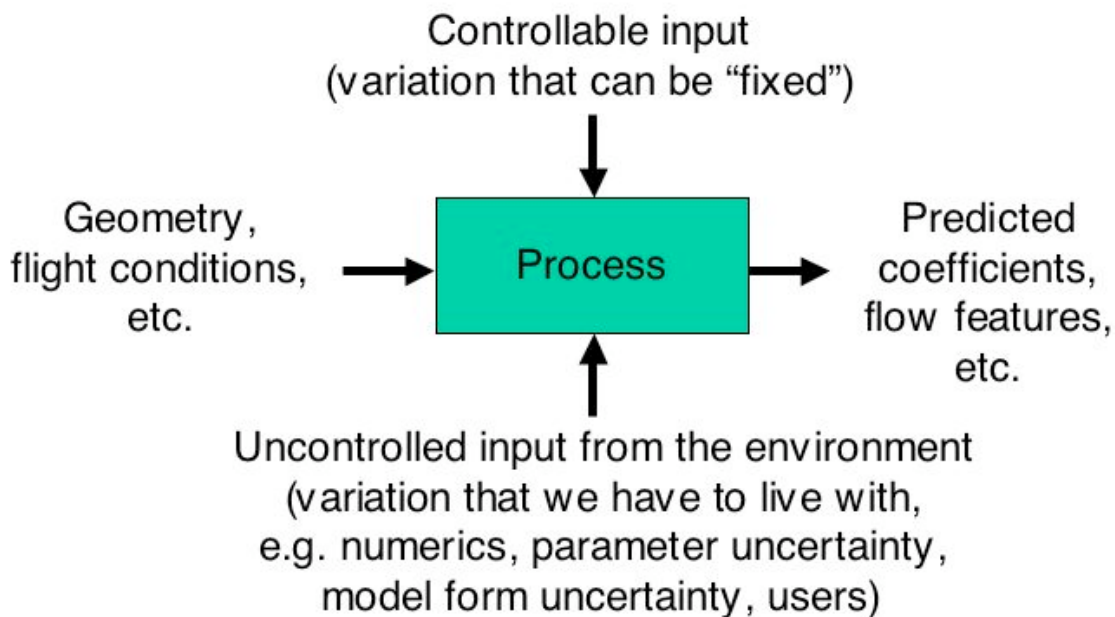
We would like to note that often the very act of measuring the outcome of a process (Evaluation) will lead to improvement in the process result. This was evident in the improvement of the Second Drag Prediction Workshop results over those of the First. We have also seen this in our development of statistical control of wind tunnel measurement processes.

If we think of prediction as a manufacturing process, then we have the situation described schematically above. We would never expect every widget coming off the line to have identical dimensions and, similarly, we should not expect every prediction to have no variation across, codes, grid types, users, turbulence models, etc.

We do want to emphasize that to realize the full benefit of thinking this way and making it happen, it will be necessary to be fairly proficient at some basic statistical methods. The methods of greatest interest are the same ones used extensively in metrology and experimentation, particularly statistical quality/process control. Fortunately, these methods are not complicated. They do, however, require the user to get into a “statistical frame of mind” in order to use them effectively and correctly.



Creating a predictable process ...



[Danger! Danger! Statistics will have to be learned and used.]

In this presentation, we talk a lot about processes because the notion is fundamental to quality assurance, especially quantitative quality assurance.

The best way that we know of to enable determination of quality is to think of computation as a process for manufacturing numbers. One of the biggest advantages of thinking this way is that we can borrow most of the methods and strategies of the manufacturing quality assurance community that have been developed over the last 80 years. In addition, we can borrow the extensions of those ideas to precision experimental work that have been developed at the National Bureau of Standards over the last 40 years.



Critical levels of attainment for a predictable process

- A defined set of steps
- Stable and replicable
- Measurable
- Improvable

The quality assurance levels listed in the slide have been implicit in the quality literature but they were first promoted heavily by the Software Engineering Institute. (see reference on slide 11). These aspects are crucial for the credible **prediction** of computational uncertainty. The DoD actually has a process for certifying the quality assurance level attained on a sustained basis by a contractor's software development process, ranging from Level 1 (no process) to Level 5 (all of the above attributes are included).

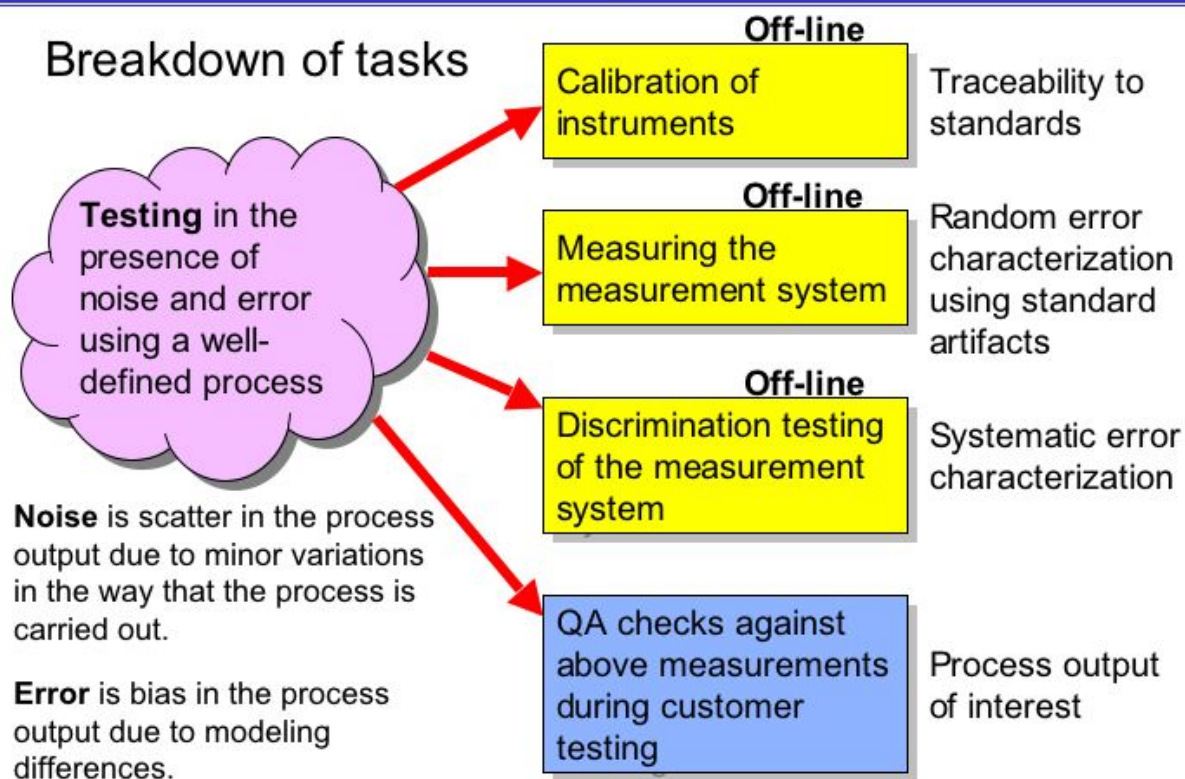
This breakdown of tasks was established by the National Bureau of Standard over 40 years ago for precision measurement in standards and calibration labs. It makes a seemingly impossible task not only tractable but controllable and credible.

The first task, Calibration of Instruments, is done offline and provides a common reference state for all facilities which are traceable to national standards.

The second task involves periodic offline testing of the measurement system using standard artifacts which are called check standards. This task is done solely for the purposes of tracking any possible drift in the mean or dispersion of the measurement output of the system. It also allows the credible characterization of that dispersion.



How it's done for an experimental process



The third task involves the off-line determination of systematic errors in the measurement system. For a wind tunnel, some examples would be imperfect knowledge of the test section calibration coefficient and imperfect correction of wall effects.

The fourth task involves those quality checks to be conducted during a test. Those checks are conducted by comparing data taken during the test for that purpose against historical data. There are many such checks that need to be done in a wind tunnel test.

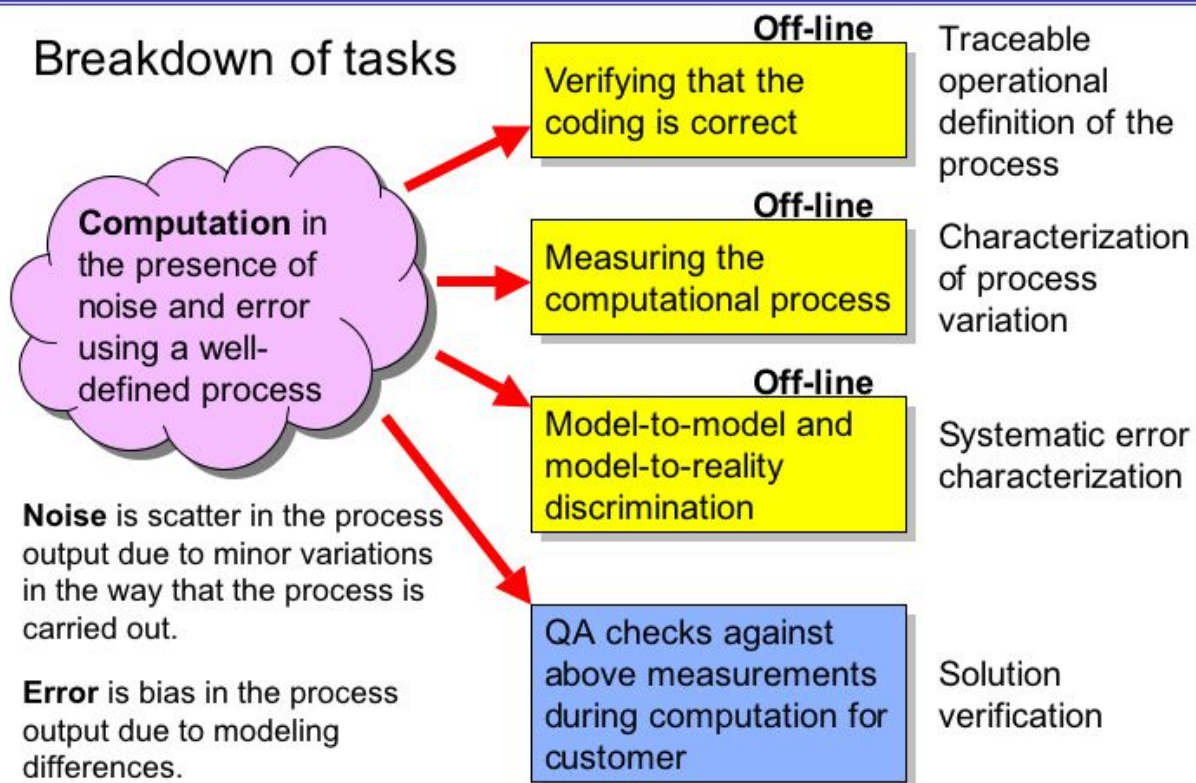
The first task again provides referenceability by being able to prove that the code is doing what it is purported to do. This task is usually called “Code Verification” (Roache)

The second task requires that the output variation of the computational process be controlled and evaluated. This can be effected through best practices and comparing the results of multiple codes, grid types, turbulence models, users, etc. There is a belief that attempts at grid convergence will be helpful here with part of this variation but preliminary results are not encouraging. This task is part of what is usually called “Solution Verification” (Roache).

The third task involves parameter and model form uncertainty. There are a variety of ways to propagate parameter uncertainty into the code output and we are encouraged that these methods not only work but



How it could be done for a computational process



can be reasonably implemented. Model form uncertainty is another story and much work needs to be done here. The most promising notion that we’ve seen is the idea from statistics of “severe testing” in which one attempts to find both the portions of the envelope where the predictions are reliable and the accuracy can be evaluated and the boundaries of those portions where the predictions become less reliable and accuracy becomes more difficult to predict. This task is usually called “Validation” (Roache).

The fourth task involves checks to be made when a prediction is being made for a customer. Here it will be necessary (1) to assure that the best practice system is being followed so that predictions of process uncertainty have credibility and (2) to estimate the locations of the envelope boundaries where the credibility of the predicted systematic uncertainty becomes more problematical. This task is the on-line part of Solution Verification.

See <http://aaac.larc.nasa.gov/tsab/cfdlarc/aiaa-dpw/>.



Recommended actions for COMSAC near-term

- **Establish a working group like the AIAA Drag Prediction Workshop (DPW)**
 - Steer activities that can be started right now
 - Select a **small** number of COMSAC focus problems
 - Use those problems
 - » to demonstrate the prediction uncertainty strategies we've proposed
 - » to find out just how tough this problem really is
- **Some useful things to do right now for estimating uncertainty**
 - Run multiple codes, different grid types, multiple turbulence models, etc.
 - Stick to realistic S&C problems, i.e. work data sets that fully capture the physics of the problem of interest.



Recommended actions for COMSAC far-term

- **Some key things to do right now to develop a powerful and reliable uncertainty quantification (UQ) process**
 - Help us establish a reasonable UQ process.
 - » Help us develop best practices and find ways to control and evaluate them.
 - » Help develop and implement tools for propagating parameter uncertainty and IC/BC uncertainty into the coefficients of interest.
 - » Help develop experiments to determine our ability to predict uncertainty and to predict the domain boundaries where the physics changes (and, therefore, probably the uncertainty).

We do not want, with the emphasis of this slide, to inadvertently give the impression that only on-line work counts. To the contrary, Slide 15 shows that we consider the off-line work described therein to be essential for a tractable and accurate process. By “local”, we simply mean local in the physical inference space (right physics).



Final (can't resist) slide

- Validation for aerodynamics is **not** a global process
 - It is a never-ending local process and depends every time all the time on the flow you are working.
 - We can greatly improve, speed up and generalize (somewhat) the process we're successfully using right now.
 - The key is quantitatively predicting the error.

The best practices work is being accomplished under NASA Langley Research Center contract number NAS1-03053.

Technical Monitor:

J. H. Morrison

757-864-2294

Joseph.H.Morrison@NASA.gov

A BEST PRACTICES SYSTEM TO ENHANCE CFD USE IN STABILITY AND CONTROL APPLICATIONS

Michael R. Mendenhall
Nielsen Engineering & Research

Reference: Contract No. NAS1-03053



COMSAC Symposium
NASA Langley Research Center
23-25 September 2003

Successful use of CFD to provide aerodynamics for stability and control (S&C) applications will require that the traditional time and costs associated with CFD be reduced and that the errors and uncertainties currently associated with CFD be better understood. CFD will be required to work under a wide range of flow conditions and provide fast and reliable aerodynamics if it is to contribute to this next generation of S&C analyses. CFD solutions have errors and uncertainties due to poorly converged solutions, solution anomalies caused by grids, turbulence models, and parameter selection, and other manifold reasons.

In addition to the above problems, there will be a requirement for communications between the CFD expert and the S&C expert and possibly experts from other related disciplines. The CFD



INTRODUCTION

- Successful use of CFD for Stability & Control will require:
 - Reduced time and cost of CFD runs
 - Increased reliability and accuracy of CFD results
 - Better understanding of uncertainties
 - Reduced errors and more efficient computations
 - Mutual understanding of CFD and S&C problems and requirements
 - Communication between CFD, Aerodynamics, and S&C engineers

expert may not understand the technical problems associated with S&C, and it is almost certain the converse is true.

Some problems to be anticipated in using CFD for stability and control involve the need for aerodynamic characteristics for a broad range of flight conditions. These flight conditions may include flow regimes that are very difficult to handle with CFD; for example, post stall flight and unsteady flow conditions. It may be necessary for the CFD engineer to use solutions that are not converged and glean the best possible aerodynamic characteristics from this information. It will be essential to know and understand the quality of the solution and the uncertainties that must be associated with the aerodynamic coefficients.

In addition, the large number of points required to complete an aerodynamic database will put strong demands on the stability and control budget. It will be necessary to set up and complete



PROBLEM DEFINITION

- S&C requirements embrace a broad range of flow conditions
- S&C applications require nontraditional use of CFD
 - Less-than-ideal flight conditions
 - ◆ Post stall
 - ◆ Unsteady maneuvers
 - Lack of converged solutions
 - Recognize “what is good enough?”
- S&C demands on computational aerodynamics
 - Faster setup and configuration modifications
 - More reliable solutions
 - Large databases (M, , , , geometry,)

runs faster and more efficiently to keep costs down. It will be important to eliminate incorrect runs and avoid unnecessary repeat runs. A typical aerodynamic database may require hundreds if not thousands of runs. This large quantity of data may require automated techniques to evaluate the results.

The CFD user has a critical role in current CFD processes, and the quality of computational aerodynamic results is subject to significant variability. Many problems can be traced to inexperienced users producing results with software they do not understand.¹



OTHER PROBLEMS

- Quality of CFD results depends on user experience
- Many problems can be traced to inexperienced users producing results with codes they do not understand

¹ Lee, J. R., "Certainty in Stockpile Computing: Recommending a Verification and Validation Program for Scientific Software," SAND98-2420, Sandia National Laboratories, November 1998.

NEAR is currently working to develop a system of best practices for CFD. The purpose of this work is to provide a set of user guidelines for running CFD codes to assist all users in obtaining high-quality solutions with reduced uncertainty and at lower cost.² The system includes specific guidelines for problem definition, input preparation, grid generation, code selection, parameter specification, and results interpretation. The objective of the best practices system is to ensure that all reasonable steps are taken to achieve the most accurate and reliable CFD solutions possible.



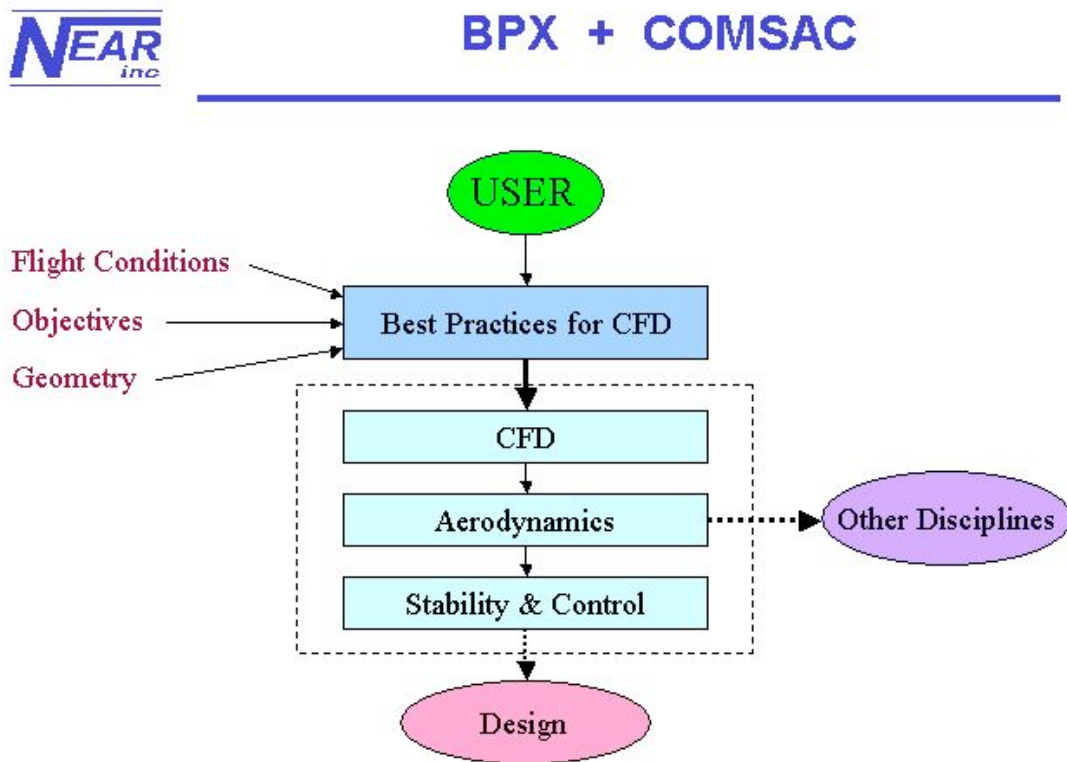
PROPOSED APPROACH

- A Best Practices System for CFD is in Development
 - Provide specific user guidelines for CFD codes
 - Ensure high-quality solutions
 - Quantify uncertainties
 - Reduce “false starts” and bad runs
 - Lower cost and time of aerodynamic solutions
- Integrate Best Practices and COMSAC
 - Automatic CFD calculations in batch mode
 - Minimal engineering intervention
 - Preliminary results checking and evaluation

² Mendenhall, M. R., Childs, R. E., and Morrison, J. H., “Best Practices for Reduction of Uncertainty in CFD Results,” AIAA 2003-0411, January 2003.

This is a diagram of a proposed approach to integrate Best Practices and COMSAC. The user will specify the flight conditions of interest, the objectives of the design, and the geometry of the configuration. Best practices will use this information, along with the expert CFD knowledge in the system, to specify a number of guidelines for setting up the CFD runs to provide the best possible aerodynamics information. These aerodynamic characteristics will be available to the stability & control area as needed for design purposes. In addition, the up-to-date aerodynamics can be made available to other disciplines in a similar manner.

In the diagram, the dashed box represents COMSAC. The aerodynamic information generated for COMSAC can be shared with other disciplines.



The objective is to provide a state-of-the-art capability for CFD analysis which can be broadly applied by users interested in (1) improving the accuracy and reducing the uncertainty of CFD results, and (2) reducing the time and cost associated with CFD applications. Best Practices must include the following:

- Provide an intuitive process for general acceptance by the CFD community.
- Demonstrate ease of use for all levels of users.
- Provide information appropriate for all CFD users to achieve more reliable CFD results with less effort.
- Provide a comprehensive compendium of procedures and expertise that should be followed to get the required accuracy from CFD.
- Evolve with advancements in CFD algorithms and codes.
- Choose a framework for best practices applicable to all algorithms and solvers that developers and/or



BEST PRACTICES OBJECTIVES

- Ensure that all levels of CFD users can obtain the highest quality CFD results possible with the methods available.
- Minimize unacceptable runs.
- Assist users in obtaining CFD solutions with reduced errors and lowest cost.
- Help users understand and estimate levels of uncertainty prior to obtaining a solution.

users are willing to support.

- Permit individual users to customize details of best practices to support specific needs and provide for proprietary versions of the system.
- Provide a self-critical system by noting the relative confidence in specific guidelines and characterizing the aspects of CFD practices that are poorly understood.
- Provide the flexibility to evolve into a future system which may require highly automated CFD quality assurance algorithms, including automatic grid generation and solution interrogation algorithms.

Since different CFD codes have strengths and weaknesses in different areas, the approach to best practices is to include information and guidelines tailored for specific codes. The knowledge database will be obtained from experienced and successful code developers and users. Links to references will provide the user with sources of additional information. A number of standard runs which can be used for validation purposes will be included. The system will be easily updatable since knowledge and experience with the codes is always changing.



BEST PRACTICES APPROACH

- Include Specific Code Information
 - Recommendations
 - Warnings
- Expert User Knowledge Preservation
 - Knowledge and experience from successful developers and users
 - Historical documents and technical references
 - Validation benchmarks and examples
- Dynamic and Updateable System
- Public and Proprietary Versions

The expert knowledge for best practices is obtained through personal interviews with expert users and developers. The recorded interviews are transcribed, checked for accuracy, and edited before the information is distributed in the knowledge database. The information is linked by a framework of keywords. The knowledge database is also linked to a references database containing citations to published information. When a keyword is selected by the user or automatically chosen by the best practices system, a search will identify the appropriate expert knowledge and related technical references.

The initial best practices system will be a public version for unlimited distribution. Future systems developed for commercial or government organizations will include proprietary



KNOWLEDGE ACQUISITION

- Basic Best Practices eXpert System (Public)
 - Interviews with expert developers and users
 - Published information and guidelines
 - Public domain examples
- Future BPX System (Proprietary)
 - Sensitive/Limited information (Corporate Memory)
 - Specific experiences and anecdotes
 - Competitive information
 - Project results

information. These systems will contain the corporate memory of the organization, and it may also include sensitive and competitive information for restricted use.

A hierarchical keyword structure is used to organize the knowledge stored in the databases. Each node in the hierarchy is represented by a keyword that describes a topic. This model allows the information stored to be linked in a logical fashion, which assists in both the knowledge acquisition and the knowledge retrieval mechanism. The user can add additional keywords as needed.

For purposes of this discussion on best practices, a high-level hierarchy of keywords is shown above. This list is just the major topic areas, and the order shown is not important. The user can enter the system at any location in the hierarchy.



BEST PRACTICES GUIDELINES

- Problem Definition
- Code Selection
- Input Preparation
- Grid Generation
- Parameter Specification
- Solution Evaluation
- Results Interpretation

The hierarchy is a way to organize the information in the databases so that it is easily accessible for editing and maintenance purposes. It is important that the keyword list be comprehensive and as complete as possible; however, the actual position or location of the topics in the hierarchy is not critical to the best-practices process. There are many links and connections between keywords at all levels in the hierarchy so that the interdependence between topics is maintained without regard to their physical position in the hierarchy.

Although the best practices system currently under development is aimed at the CFD user, it will be equally usable by technical managers or the engineers in S&C (or other discipline) to better understand the problems and difficulties associated with achieving good CFD results. It is conceivable that a future system which includes expert knowledge and experiences from S&C or other discipline could be equally useful to the CFD engineer. A future COMSAC capability could include a coupled system of best practices and expert knowledge from S&C, CFD, and other related disciplines.



USE OF BEST PRACTICES

- Code Developers
 - Identify specific problem areas
 - Understand uncertainties
- Design Applications
 - Provide experience (new code, new engineers,)
 - Assist at all levels of use
- Technical Managers
 - Understand technical issues and problems with CFD results
 - Quantify uncertainties and potential errors
 - Understand CFD capabilities and quality of results

A future BPX/COMSAC system could include an expert knowledge database with stability and control knowledge and prediction methods. This would be a way to preserve the corporate memory in stability and control for use by future generations of engineers. The advantages of having this information available for training purposes and for use in future designs are manifold. First, it is a way to reduce cost and risk on future programs by eliminating the mistakes that have already been made. Second, it provides a way to train the new engineers who may not have access to the senior engineers who did the original work. Finally, it will maintain the organization experience base as engineers retire or otherwise become unavailable to the technical discipline.



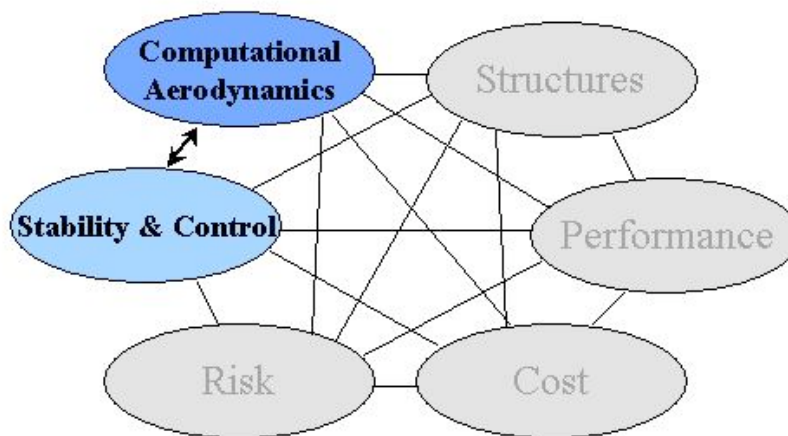
FUTURE BPX/COMSAC SYSTEM

- Include S&C Knowledge and Methods
- Integrate CFD, S&C, and Related Disciplines
- Advantages
 - Reduce risk and development costs
 - Stimulate communication between disciplines
 - Minimize uncertainties in preliminary and final designs

This chart illustrates a future integrated design and analysis system with several related disciplines shown for example. The computational aerodynamics discipline will be run by a best practices system described previously. Similarly, the stability and control discipline could have its own best practices system as suggested on the previous chart. The two disciplines could be linked by an overall best practices framework that will permit them to work together efficiently. The complete system could be expanded incrementally to include the other disciplines to provide a multidisciplinary expert design system.



FUTURE INTEGRATED SYSTEM



The approach of coupling expert knowledge and prediction capability has been demonstrated successfully by NEAR in other technical areas. LVX is a system for the aerodynamics of launch vehicles which couples expert knowledge, corporate memory, design experience, and aerodynamic prediction methods.³ RSX is a similar system based on the same computational framework for the aerodynamic design and analysis of rocket sled test vehicles.⁴



CURRENT SYSTEMS

- Current working systems can be leveraged for COMSAC
 - Ensure success at reduced risk
 - Lower development cost
- Existing integrated design systems (at NEAR)
 - LVX - Launch vehicle aerodynamic design and analysis
 - RSX - Rocket sled aerodynamic design and analysis
 - BPX - Best Practices for CFD
 - CFDX – CFD Exchange

³ Mendenhall, M. R. and Hegedus, M. C., "LVX – An Integrated Aerodynamic Design and Analysis Method," ICAS 2002-0234, September 2002.

⁴ Mendenhall, M. R. and Hegedus, M. C., "An Engineering Analysis Tool for Rocket Sled Aerodynamics," NEAR TR 582, February 2003.



CONCLUSIONS

- Best Practices eXpert System
 - Feasible for use in COMSAC applications
 - Potential to increase efficiency of CFD solutions
 - Means to understand and quantify uncertainties
 - Possible leveraging of previous work
 - ◆ Reduce risk
 - ◆ Lower cost
- COMSAC Will Require a Best Practices Tool to be Useful, Reliable, and Practical
- Extension to Other Disciplines Possible

Dynamic Water Tunnel Testing for Code Benchmarking

COMSAC
NASA Langley Research Center

September 23-25, 2003

AeroArts LLC, PO Box 2909

John.Hodgkinson@AeroArts.com

Palos Verdes Peninsula

Brooke.Smith@AeroArts.com

310 791 7446

CA 90274-2909

310 548 0927

AeroArts proprietary rights are included in the information disclosed herein. By accepting this document, recipient agrees that neither this document nor the information disclosed herein, nor any part thereof, shall be reproduced or transferred to other documents or used or disclosed to others for any purpose except as specifically authorized in writing by AeroArts.

This picture not only shows the first flight of an aircraft, it also shows the first flight test. The Wright brothers' experience in this flight test has been repeated in almost every new aircraft program to this day. That is, the brothers discovered aerodynamic/handling qualities/flying qualities anomalies that were not predicted by ground based tests and calculations.

However, discovery of such problems in the flight test phase of an aircraft development is too late.

- Research is difficult and expensive,
- Experiments are challenging,
- Budget and Schedule pressures are high,
- Configuration changes are costly,

Dynamic Water Tunnel Testing



AeroArts LLC

Copyright © 2003 AeroArts LLC All rights reserved.

030925- 2

S&C Anomalies

Discovery of S&C anomalies while in the flight test phase of an aircraft development is too late.

- Research is difficult and expensive
- Experiments are challenging
- Configuration changes are costly
- Budget and Schedule pressures are high
- Fixes are limited



Introducing Dynamic Water Tunnel Testing

False color flow visualization of a generic transport model at static flight condition.



At 1 foot/sec in water, Reynolds number is $\sim 100,000$ per foot. Skin friction drag will be incorrect. Boundary layers will separate prematurely on curved surfaces. Tripping is not practical. Don't use a water tunnel for designing airfoils or determining transport performance. However, for many of the "interesting" parts of the envelope where non-linear or unsteady aerodynamic phenomena exist, even the full scale aircraft will be experiencing massive separation. Sharp edge configurations will be even less sensitive to Re. Vortex effects, pressure footprint, trajectory, and burst points are relatively insensitive to Reynolds number. Mach number will be $\sim$ zero. However, most aircraft will not be at high speeds for very long if maneuvering.

Reynolds, Mach, Cavitation

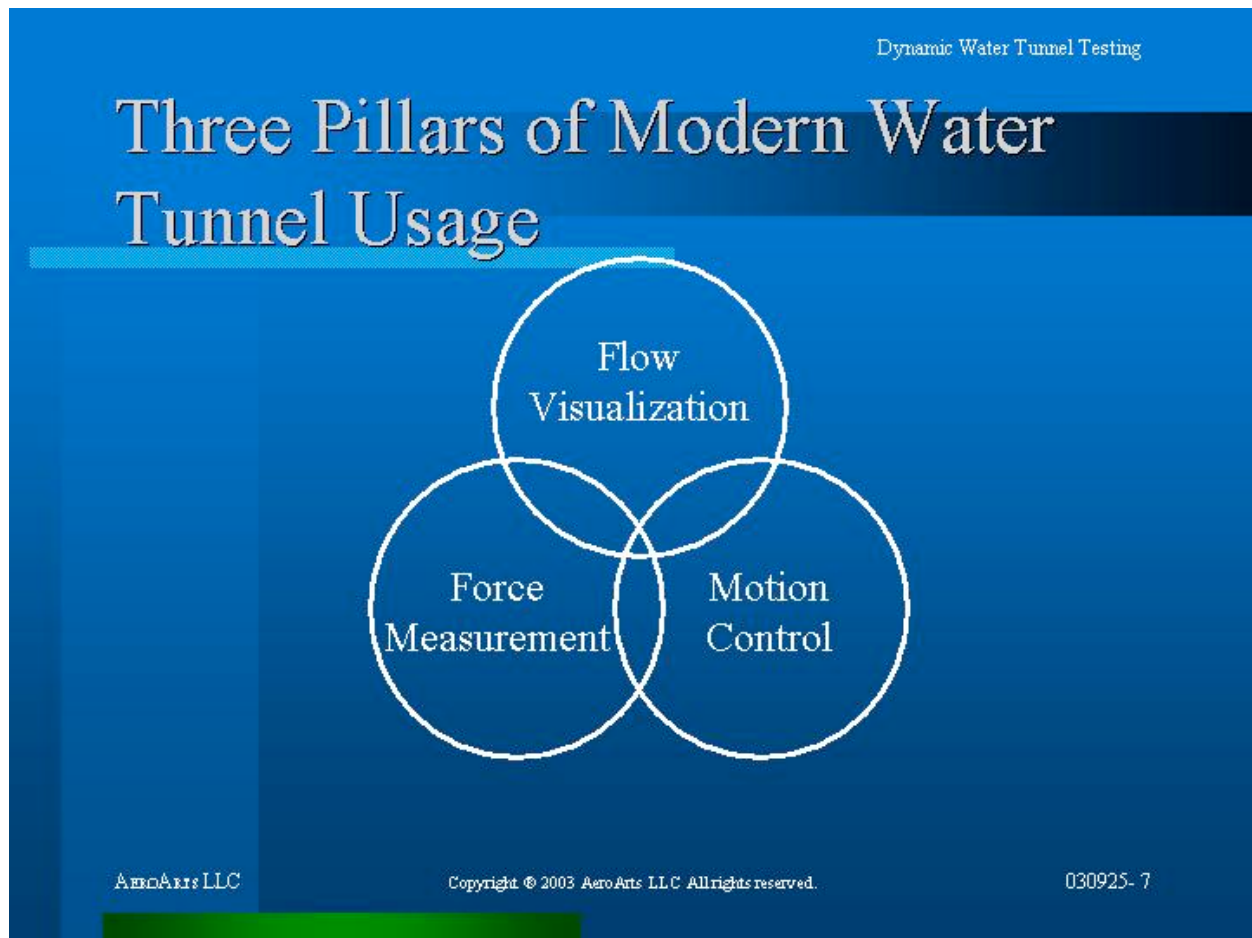
- **Reynolds number effects**
 - Drag
 - Premature separation
 - Vortex behavior / effects not effected
- **Mach number**
 - Sonic velocity in water is 4x speed in air
- **Cavitation**
 - Not an issue for low speed tests

Made a few improvements to traditional water tunnel testing – moving beyond pretty pictures.

Added a computer-controlled model support, a submersible force balance, image analysis programs, and a computer to coordinate it all.

We think it is an ideal environment for addressing the issues of developing better predictive models. Not only as a primary research facility, but also in a supporting roll for development, validation, and verification of other predictive methods, such as CFD.

Here's why...



Dynamic scaling of motion requires that the non-dimensional angular rates or frequencies are matched.

This example compares a full scale fighter type aircraft with a typical sub-scale models tested in a wind tunnel. The fighter, with a flight speed of 300 feet/sec, and a roll rate of 180 degrees/second will have a non-dimensional roll rate of 0.2.

To match this rate, a typical sub-scale model in a typical low speed wind tunnel will need to roll at 1800 degrees/second, or 10 times the full scale rate. Compare this to the same test conducted in a water tunnel. Typical model scales and test velocities result in the water tunnel model rotating at 18 degrees/second.

It is convenient to think of this as time going 10x faster than real time in a wind tunnel, and 1/10x in the water tunnel.

Dynamic Water Tunnel Testing

Motion Scaling

	FULL SCALE AIRCRAFT	WIND TUNNEL ROTARY TEST	WATER TUNNEL ROTARY TEST
$k = \frac{a\dot{b}}{2V}$	0.2	0.2	0.2
V	300 fps	200 fps	8 ips
b	38 ft	2.5 ft	10 in
ω	180 deg/s	1800 deg/s	18 deg/s

AeroArts LLC Copyright © 2003 AeroArts LLC All rights reserved. 030925-8

Scaling slows down time in water

- **Because of motion similarity requirements**
- **Data acquisition easy**
- **Rotary motions slowed:**
 - 10X less than full scale
 - 100X less than wind tunnel
- **Balance, model support lighter**
- **Flow easier to visualize**
- **Real-time control more feasible**

Water tunnels are renowned for outstanding flow visualization, usually at static conditions.

Here we have added a moderate amplitude motion. Simple single-axis harmonic motion.

Good, but yields limited information – questions are still unanswered – need more data – more experiments.

Dynamic Water Tunnel Testing

Dynamic Flow Visualization

- Moderate amplitude, 30° , body axis roll oscillation
- 30° pitch attitude



AEROARTS LLC Copyright © 2003 AeroArts LLC All rights reserved. 030925- 10

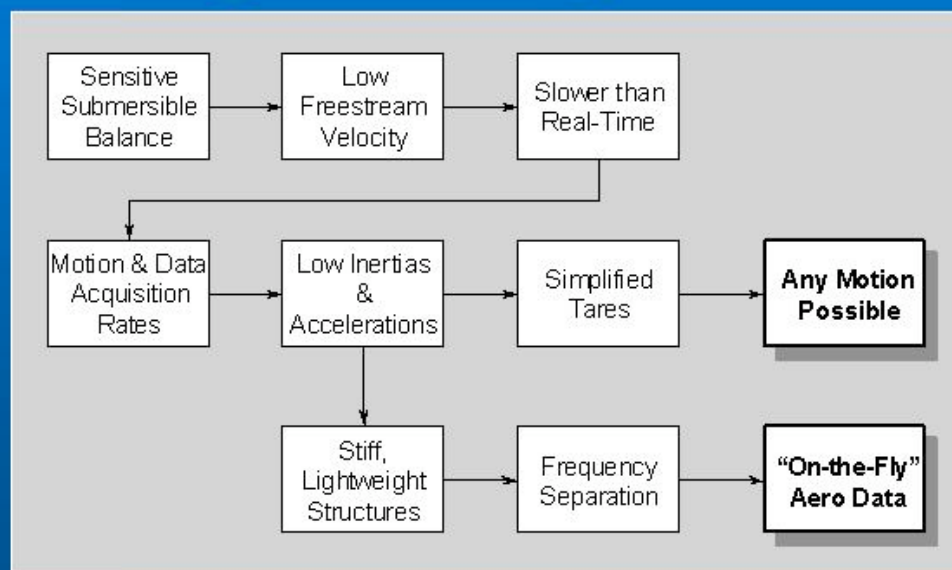
Time runs slower in the water tunnel test. Accelerations and inertia forces scale with time squared.

As a result, inertia forces developed by a maneuvering model are very small ($\sim 1\%$) compared to the aerodynamic forces of interest, and can be lumped into an error budget.

Fluid forces dominate in water

- **Inertial tares negligible**
 - Balance highly accurate
 - Arbitrary motions easy
- **Slow-motion model support structure**
 - Far less massive than for testing in air
 - Support frequencies far greater than model frequencies

Dynamic Testing in Water: Enabling Technology Chain

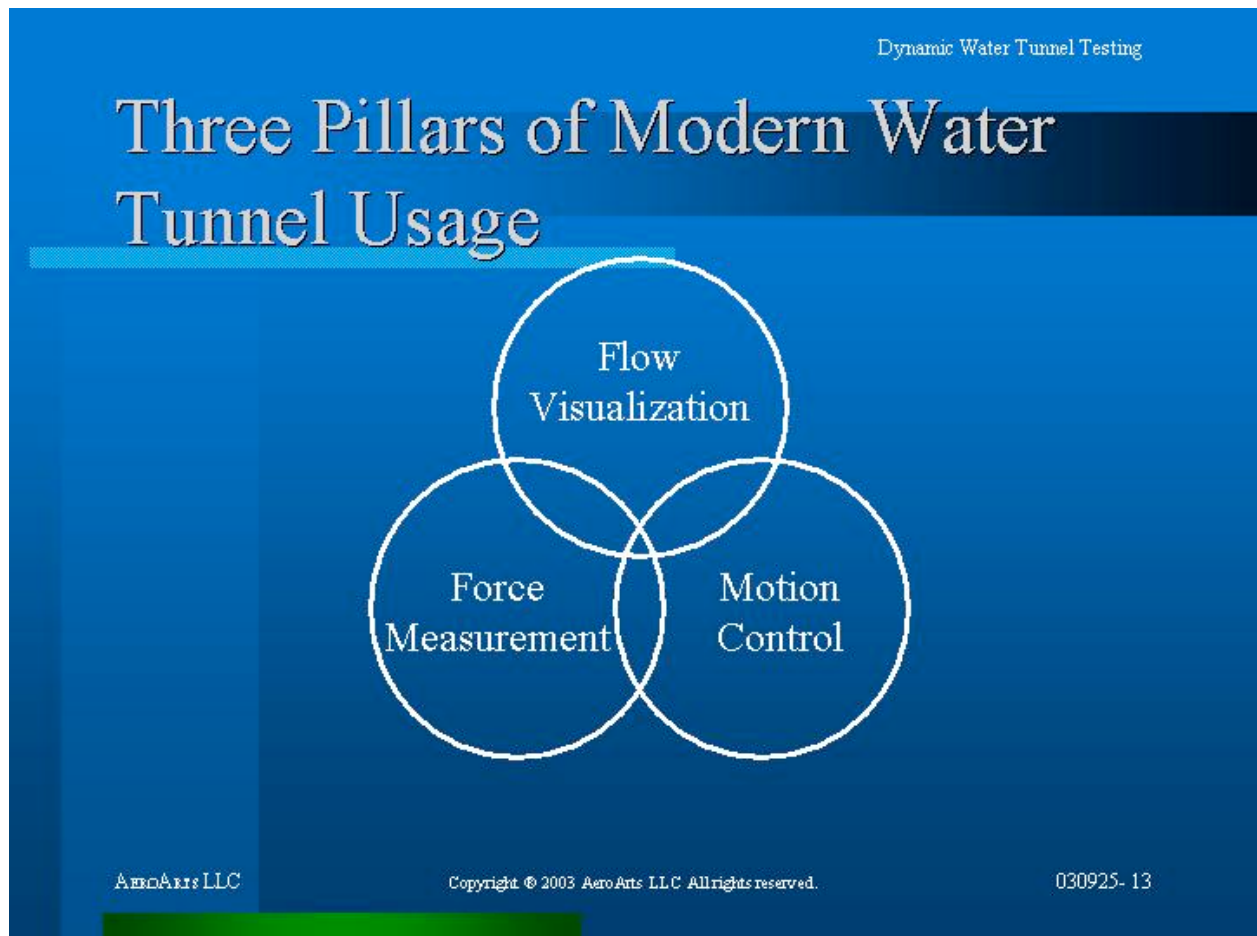


This is where it all comes together, the three pillars:

Advantages:

- Outstanding flow visualization
- Force measurement
- “Unlimited” motions possible
- Self-consistent tests -- Assumption Checking
- Separation of structural and aerodynamic frequencies
- Simple dynamic tares

Possible to test any motion or maneuver.





Introducing Scorpio

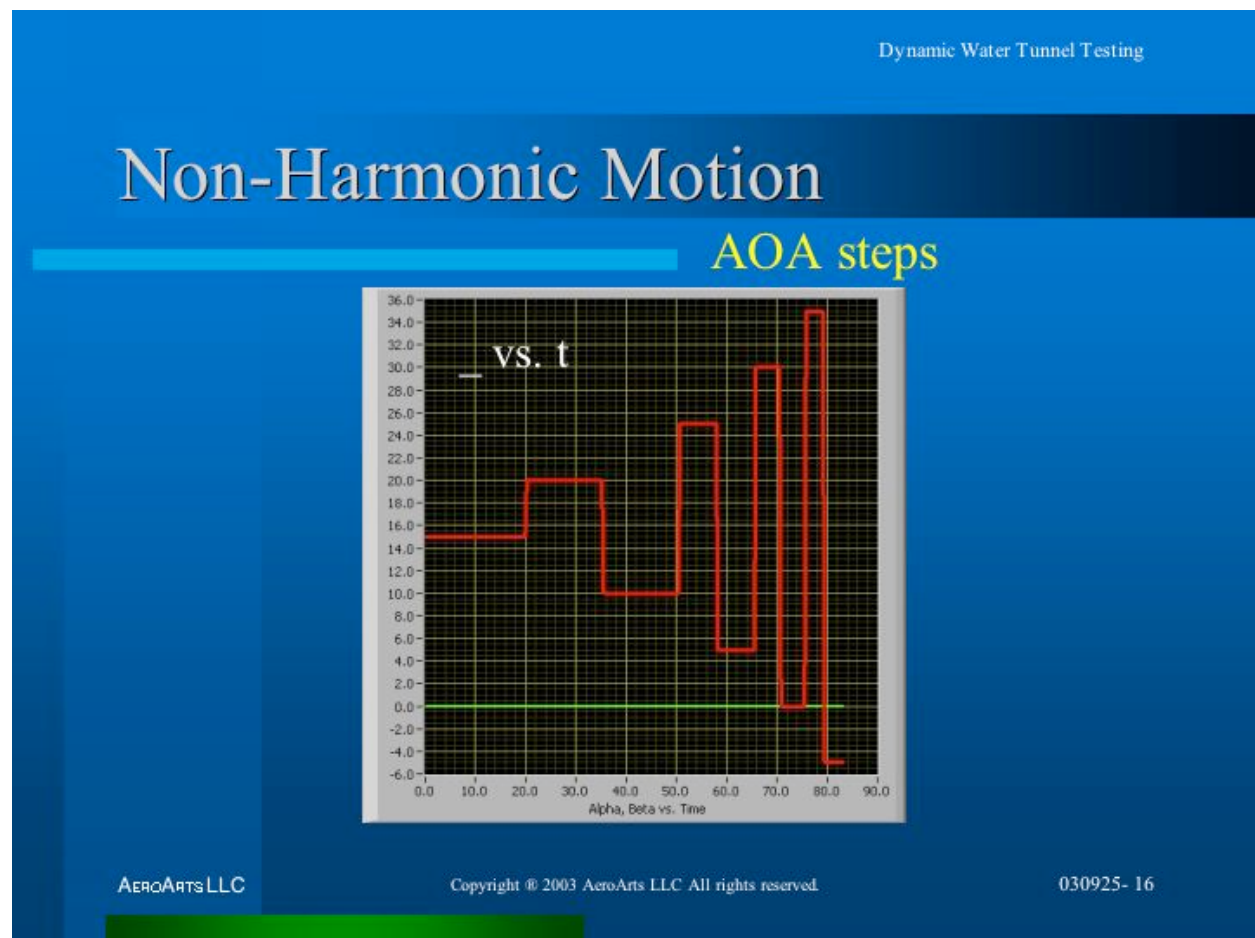
AeroArts Scorpio System - Highlights

- **Model Support:**
 - 6-DOF closed-loop servo under computer control
- **Force/Moment Balance:**
 - Miniature, submersible 6-component
- **Flow Visualization:**
 - Integrated with data acquisition and motion control
 - Automated image analysis and feature extraction
- **Operational Software:**
 - National Instrument's LabView
 - Complete, integrated test environment

As a benefit of the stiff structure and low mass, the system is not limited to harmonic motions (sine, cosine). Here is a demonstration of pure angle of attack steps, by performing triangular wave z translations with increasing rates. The final reversal in this example executes a 40 degree AOA step.

AOA initial 15° , delta AOA = $\pm 5^\circ$, $\pm 10^\circ$, $\pm 15^\circ$, $\pm 20^\circ$

In addition to standard pitch motions that have $\text{alphadot} = \text{pitch rate } (q)$, pure alphadot , pure q , or any combination thereof can be performed. Likewise, betadot can be specified independent of r (yaw rate).

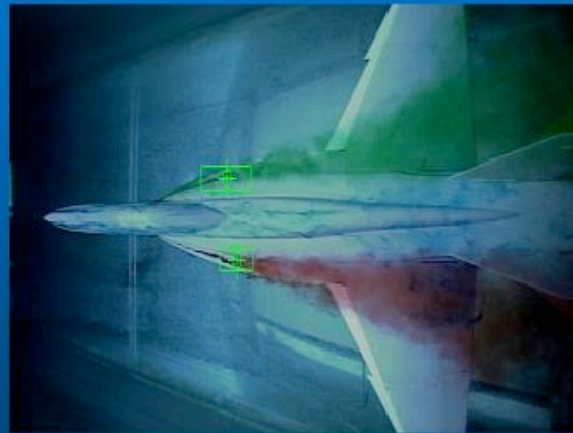


A recent test performed by AeroArts used the Scorpio data acquisition and image capture system to grab flow visualization photos concurrently with force and moment data. Custom image analysis algorithms automatically isolated the left and right LEX vortices, identified the vortex burst points as shown in this screen capture. The software recorded the images, the burst points, and plotted the burst point migration in real time as the test was being performed.

Dynamic Water Tunnel Testing

Quantitative Dynamic Flow Visualization

- λ **Automated flow feature detection**
- λ **Maneuvering model**
- λ **Quantified burst location**
- λ **Concurrent force measurement**



AEROARTS LLC

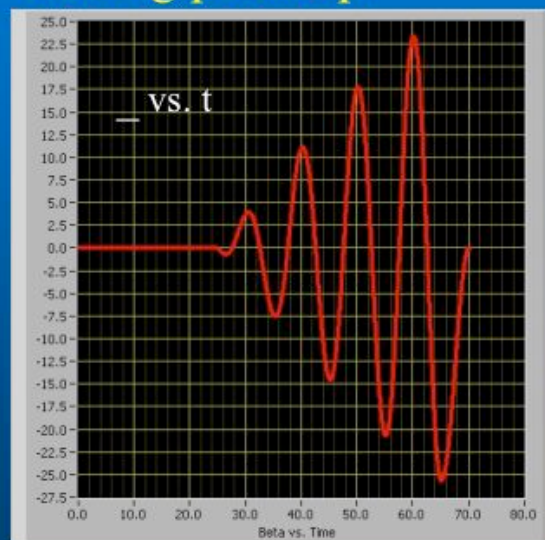
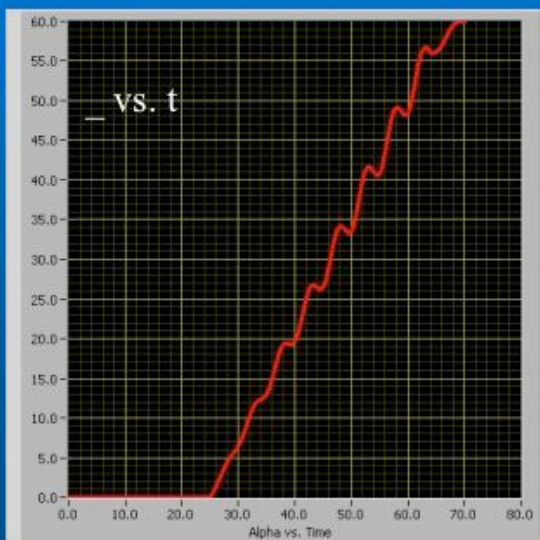
Copyright © 2003 AeroArts LLC All rights reserved.

030925- 17

Multi-axis motion – a rolling pitch-up.
 $\pm 30^\circ$ body axis roll with a pitch ramp from 0° to 60° .

Multi-Axis Motion

Rolling pitch-up

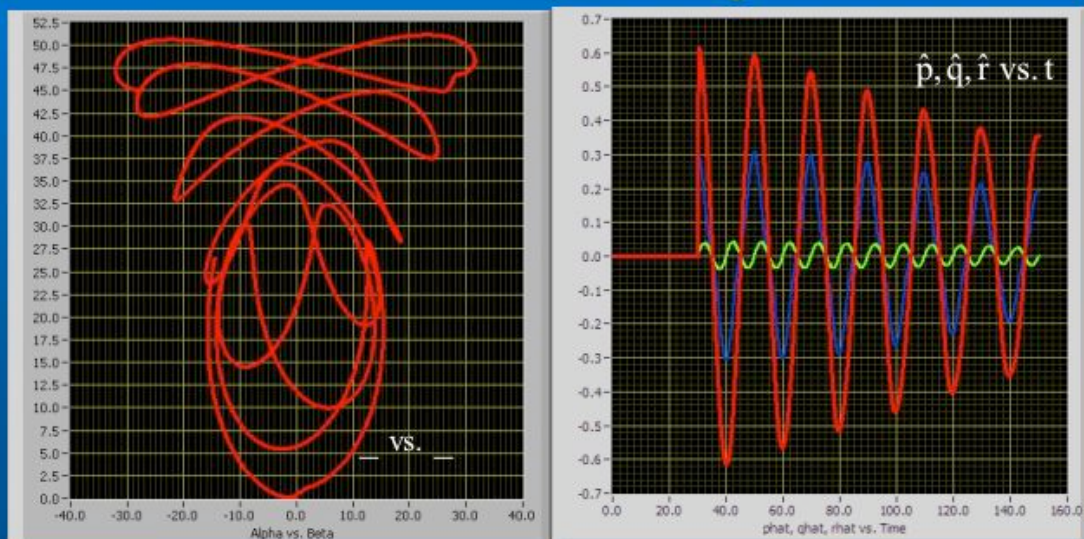


A falling leaf-type maneuver idealized for study from an in-flight incident. The falling leaf incident began with a roll reversal at a low angle of attack and developed into a roll and yaw oscillation with a sinking flight path.

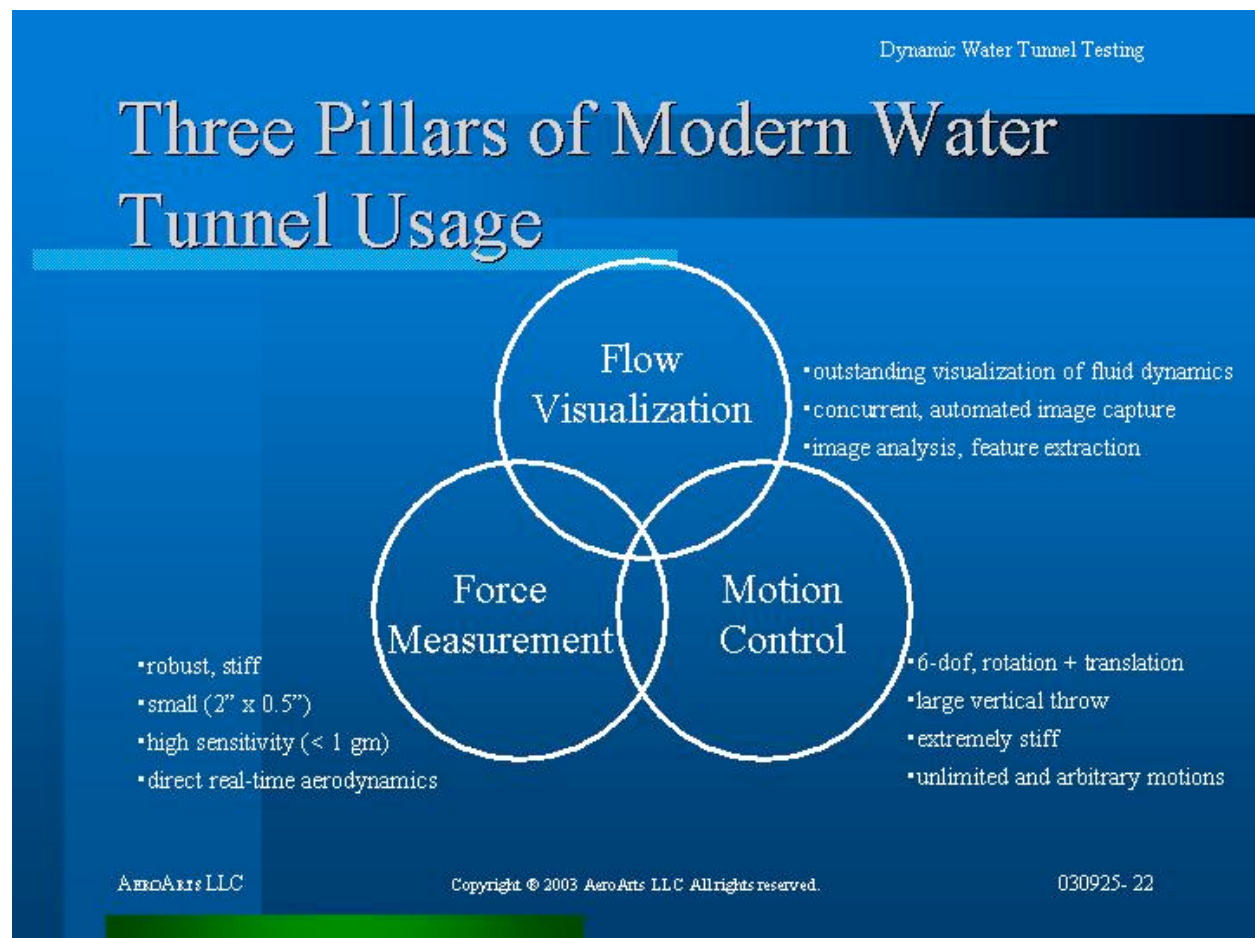
This motion requires rotations in all axes and coordinated translations in all three axes.

In-Flight Event (6 dof)

Falling leaf



Here's the tool. What can it do for you?



Adaptable

Any motion (that doesn't crash into something) is possible. Powered rotorcraft models have been run with full 3-axis swashplate control under computer control.

Dynamic Water Tunnel Testing

Dynamic Water Tunnel Testing for Code Benchmarking

- **Adaptable**
 - Any motion possible
 - Powered models
 - Active control surfaces
- **Proficient**
 - Accurately produces desired motions
 - Reveals fluid dynamics
 - Captures nonlinear and time-varying aerodynamics
- **Cost Effective**
 - Models are relatively inexpensive
 - Facility time is cheap
- **Calibrate and Validate in consistent environment**
 - One model, one facility, many tests

AeroArts LLC Copyright © 2003 AeroArts LLC All rights reserved. 030925- 23

Intent of this talk is to present the S&C priorities as seen by the Langley team. No roadmaps or 5 year plans will be presented. We are actively soliciting your feedback, your ideas, and your help in building and executing this program.

COMSAC: Visions and Potential Program Content

Robert M. Hall and C. Michael Fremaux
NASA Langley Research Center

Joseph R. Chambers
ViGYAN

*COMSAC Symposium
September 25, 2003*



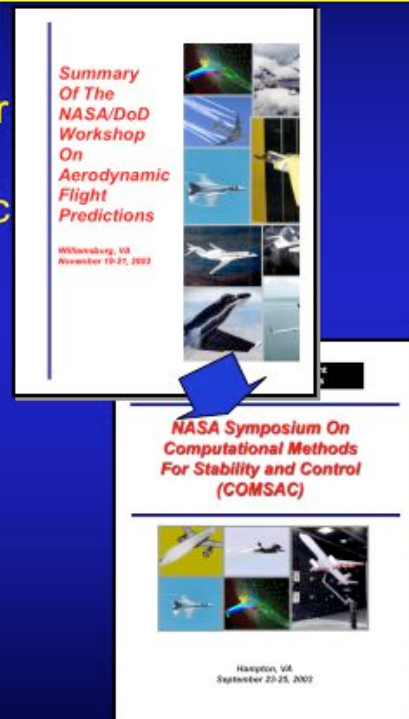
Outline

- Background
- NASA constraints and priorities
- Potential program content
 - High priority issues
 - Approach
- Prepared critiques
- Comments by attendees
- Closing comments



Background

- Promise of CFD highlighted by Abrupt Wing Stall program and other activities
- Needs apparent during Aerodynamic Flight Prediction Workshop
 - Williamsburg, VA, Nov. 19-21, 2002
- Industry/DOD tour
 - NAVAIR 2/11/03
 - Boeing Seattle 4/28/03
 - Lockheed-Martin Ft. Worth 4/29/03
 - Boeing St. Louis 4/30/03
 - Lockheed-Martin Marietta 5/2/03
 - AFRL 6/5/03
- COMSAC Symposium





Preliminary Feedback

- Much skepticism on CFD applications to S&C (both S&C and CFD communities)
 - Considerable “wait & see” attitude
- Only “9-1-1” activities in most organizations
- Limitations of current tools acknowledged
- Some potential collaborative efforts identified
- NASA encouraged to continue planning



NASA Constraints & Priorities

- NASA guidance for research on military configurations
 - OMB: Cooperative work with DOD encouraged when there is a dual civil/military application
- NASA focus is generally on civil configurations
- COMSAC planning continues to identify civil and military issues





COMSAC Objective

- Accelerate the application, validation, and focused development of CFD methodology to S&C aerodynamic predictions and analyses
- Payoffs:
 - Better understanding & control of flow physics
 - Reduced and focused wind-tunnel and flight tests
 - Risk reduction while reducing costs
 - Fewer surprises in flight test & certification
 - Minimizes “cut and try” efforts in flight test



COMSAC Team

- Programmatic responsibility: Jim Pittman (EASI lead)
- Former programmatic responsibility: Long Yip
- Technical lead: Bob Hall (S&C, CAB)
- Technical co-lead: Mike Fremaux (S&C, VDB)
- CFD lead: Paul Pao (CAB)
- CFD consultant: Jim Thomas (CMSB)
- S&C: Joe Chambers (Consultant)



COMSAC Team, Concluded

- CFD: Bob Bartels (AB)
- CFD: Bob Biedron (CMSB)
- CFD: Neal Frink (CAB)
- CFD: Farhad Ghaffari (CAB)
- S&C: Larry Green (MDOB)
- S&C: Pat Murphy (DCB)
- S&C: Ray Whipple (VDB)

People under Contract or Grant:

- CFD: Jim Forsythe, COBALT Solutions
- CFD: Case van Dam, UC Davis



COMSAC CFD Strategy

- Establish S&C “benchmark” cases to calibrate & validate computational tools for most pressing problems
- If wind tunnel or flight data are insufficient, conduct required experiments
- Major thrust is to take off-the-shelf RANS or RANS+ codes and apply
- Codes **MUST** be run “blind.” Data revealed after predictions.
- If codes are inadequate, **focused** code development for S&C applications will be pursued
- Secondary thrust is to assess accuracy of reduced-physics, engineering codes
- Balance between generic and specific applications

A program like COMSAC will have to have a balance of generic and real configurations.



Generic and Real Configurations

- Generic configurations
 - + Isolate flow physics of interest
 - + Reduce resource requirements through simplified geometry
 - + Eliminate proprietary constraints
 - + Facilitate academic/small industry involvement
 - + Can facilitate instrumentation/diagnostics
 - Usually lacks flight validation data
- Real configurations
 - + Provide access to flight validation
 - + Offer "ultimate" test because of complexity
 - + Needed to convince S&C community
 - Include proprietary issues





COMSAC General Issues

- Streamlined CFD analysis process is overarching need
 - Automated
 - User friendly
 - Robust
- Another technology issue is being able to simulate laminar separation bubble and turbulent reattachment
- How valuable are correct answers?
 - RANS vs RANS+ (DES)



Prioritized S&C Issues NASA Perspective

- High lift S&C
- Dynamic derivatives
- Wing stall progression, unsteadiness, and hysteresis
- Flow control devices
- R_n and M_∞ effects
- Hinge moments, loads
- Aeroelastic effects
- Cruise S&C
- High- α or upset conditions
- Accurate predictions for trim
- Interactions between closely coupled control surfaces
- Ground effects
- Propulsion-induced effects
- Wind-tunnel operational effects



Prioritized S&C Issues Mentioned in Symposium

- Longitudinal stability (including C_{m_0})
(Donaldson)
- High- α lift curve definition
(Killingsworth)
- Prioritized list by Bogue
- C_m as a function of α for DC-9 and F-16
(Mason)



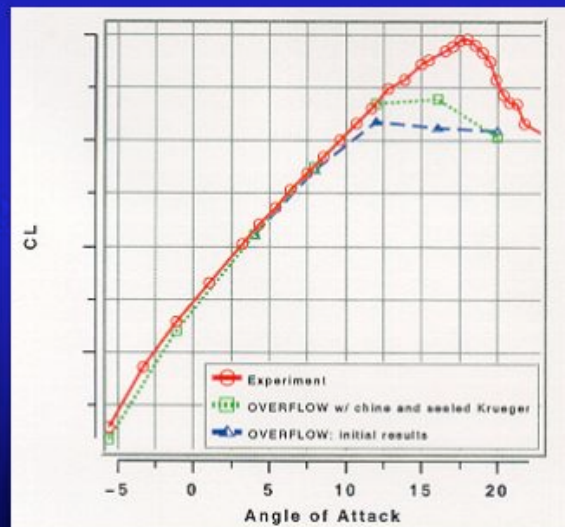
High Lift S&C for Civil Aircraft

The Challenge

- Why
 - High lift phase is critical to aircraft operations
 - SOA for CFD applied to high lift is similar to that for performance problems 10 or 15 years ago
- Example

$C_{L,max}$ and local slopes are typically missed

Data courtesy of David Bogue. Citation is Rogers, Roth, Cao, Slotnick et al., "Computation of Viscous Flow for a Boeing 777 Aircraft in Landing Configuration," AIAA 2000-4221





High Lift S&C for Civil Aircraft Notional Approach

- Potential barriers
 - Complex geometry drives gridding requirements
 - Flap elements dominated by transition and separation issues, gap flows, multi-element interactions
 - Turbulence modeling
 - Role of transition, laminar separation bubbles
- Near-term plan
 - Identify and assess data bases
 - Extend Ground-to-Flight Scaling tests of a Boeing configuration to sideslip
 - Assess 3 or 4 code systems against best available data
 - Flow physics
 - Grid adaptation strategies
 - Algorithms and turbulence modeling
 - Decision point/milestone--are additional data required?
 - Workshop in 2 years time to report out and direct direction/needs



Trap Wing
in 14x22



777 in NTF



Dynamic Derivatives

The Challenge

- Why
 - CFD needed to augment relatively poor predictive capability
 - Limited experimental capability
 - Simplified analysis methods
 - Damping derivatives are critical
 - Predicting flying qualities near stall
 - Large impact on flight control systems

- Example

Large differences in damping between configurations can result from differences in leading edge sweep and time lags in vortical flow development →





Dynamic Derivatives Notional Approach

- Potential Barriers

- Resource requirements for unsteady calculations
- Calibration data
- Nonlinearities of derivatives



F/A-18C on
Free-to-Roll
(FTR)
Test Rig
during AWS
Testing

- Plan

- Identify suitable experimental data bases
- Extend FTR testing to 757 and BWB to complement existing 757 forced oscillation (FO) data base
- Simulate both FTR and FO motions of 757 and BWB with computational tools
- Exercise time-accurate codes to simulate unique motions to measure true α and β dot terms

757 in
Langley 14
x 22



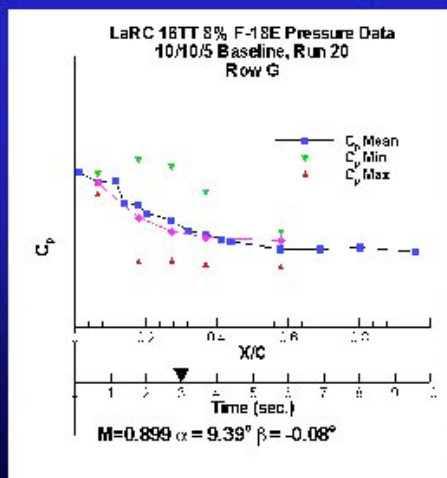
BWB





Stall Progression, Unsteadiness and Hysteresis--The Challenge

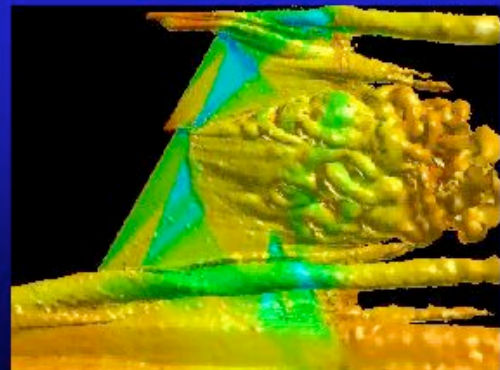
- Why
 - Stall progression characterization is critical to predicting stability near wing stall
 - Shock unsteadiness integral aspect of transonic stall
 - Interpretation of hysteresis not understood (challenge given by Dale Lorincz)



Note large shock movement during stall process for F/A-18E

Exp by Schuster

CFD by Forsythe



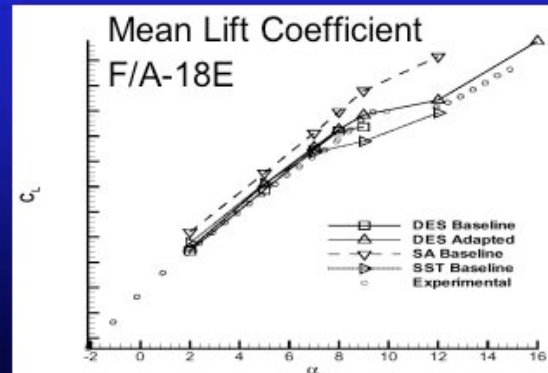


Stall Progression, Unsteadiness and Hysteresis--Notional Approach

- Potential Barriers
 - Resource requirements for unsteady calculations
 - Bistable flow states
 - Turbulence models
 - DES vs RANS
 - Grid refinement strategy for DES

Time-averaged values of unsteady solutions significantly improve correlation with data

- Plan
 - Summarize and collect data
 - While stall progression addressed in AWS program, not tackled with S&C in mind
 - Continue to study impact of hysteresis
 - Conduct workshop in 2 years





Passive and Active Flow Control The Challenge

- Why
 - Passive devices, such as vortex generators (VGs) and vortilons, are important solutions to flow control problems
 - Subscale development of VGs is problematic
 - Successful CFD characterization could reduce risk
 - Challenges more severe with unsteady, active flow control concepts
- Example

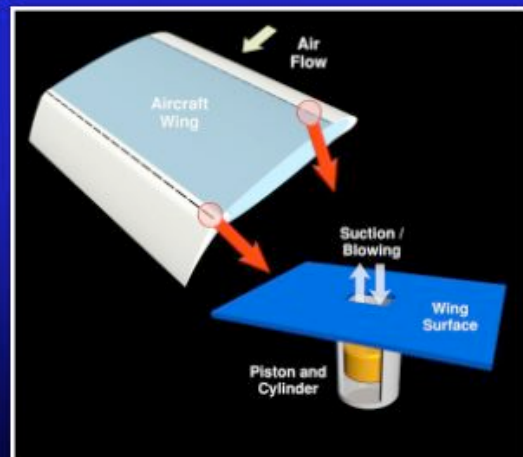
VGs and vortilon on
prototype AV-8B





Passive and Active Flow Control Notional Approach

- Potential barriers
 - Gridting requirements--complex
 - Turbulence modeling
 - Time accurate solution requirements
- Near-term plan with passive flow control
 - Capitalize on Boeing work
 - Identify and assess data bases for passive flow control devices
 - Need to examine impact of rates and sideslip
 - Assess gridding requirements
- Far-term plan--address active flow control characterization
 - CFD should lead in application strategy
 - Benchmark/Validation workshop in 04 already planned with Langley Time Accurate Program



The list is our best shot of what we view as important while we understand that there are a myriad of candidates. If an issue that is important to you is not reflected in the list, let us know!



Prioritized S&C Issues

- High lift S&C
- Dynamic derivatives
- Wing stall onset and progression, including unsteadiness
- Flow control devices
- R_n and M_∞ effects
- Hinge moments, loads
- Aeroelastic effects
- Cruise S&C
- High- α or upset conditions
- Accurate predictions for trim
- Interactions between closely coupled control surfaces
- Ground effects
- Propulsion-induced effects
- Wind-tunnel operational effects

Another technology issue is being able to simulate laminar separation bubble and turbulent reattachment



Summary of COMSAC Vision

- Have shared our philosophy
 - CFD maturity is pushing the application toward S&C, which could readily use calibrated CFD tools
 - Tackling the problem will take generic and realistic, flight configurations
 - For CFD to have an impact, it must demonstrate its predictive capability against “benchmark” cases from both the shallow and deep ends of the pool
 - Workshops to discuss specific “challenge” areas
- Have shared our vision of important S&C issues



Prepared Critiques

- NASA's vision of COMSAC options forwarded to reviewers on Tuesday
 - Pradeep Raj, Lockheed Martin
 - John Clark, NAVAIR
 - Doug Ball, Boeing Commercial
- Reviewers will follow me
- Comments and critique then taken from general audience



Open Discussion

- Opportunity to philosophize, comment, and critique what has been said
- Also seeking feedback via post-meeting evaluation forms



Closing Comments

- CFD technology and computational resources are poised to make inroads into the S&C arena
 - Can fill in the experimental gaps
 - Will be a natural complement to experiment
 - Expected to significantly reduce amount of wind tunnel and flight testing
 - Large impact on risk reduction
- Challenges are many
- Objective of COMSAC is to accelerate and focus national efforts in this area

COMSAC Feedback

Pradeep Raj
September 25, 2003

Copyright © 2003 by Lockheed Martin Corporation. All rights reserved.

Few Observations



- **Symposium has been a Worthwhile effort at improving communication among key stakeholders**
- **Laudable Objective—”...Accelerate and Focus National Effort in this Arena....”**
- **Challenges**
 1. *National Effort—how to make it “win-win”*
 2. *Focus on the “Right” Question*
 - Why S&Cers are not using CFD big time?**
 3. *Work the “Right” Issues from S&C and CFD perspective*

S&C Issues: (LM Priority)



- **High lift S&C**
- **Dynamic derivatives (2)**
- **Wing stall onset and progression, including unsteadiness (1)**
- **Flow control devices (2)**
- **Rn and M_{∞} effects (1)**
- **Hinge moments, loads (2)**
- **Aeroelastic effects (2)**
- **Cruise S&C**
- **High- α or upset conditions (1)**
- **Accurate predictions for trim (2)**
- **Interactions between closely coupled control surfaces**
- **Ground effects**
- **Propulsion-induced effects**
- **Wind-tunnel operational effects**

Streamlined CFD analysis process is overarching need

- Automated
- User friendly
- Robust

“Walk Before We Run”

Lockheed Martin Aeronautics Company

CFD Issue: Maximize Effectiveness



- **Quality**
 - All “Key” Geometric Details and Flow Conditions
 - “Real” Flight Conditions
 - Aeroelastic Deformation
 - Power Effects
 - Complete Flight Envelope (Static and Dynamic)
 - Approach to Developing and Implementing “Validation Plan” for Each Application
 - Cannot Create “Validated Code” for ALL Applications!
- **Acceptance**
 - Ease of Use
 - Short Turnaround Time (Each Analysis and Complete Dataset)
 - Low Computational Resources (Time and Memory) and Labor Hours
 - Provisions for Software Portability, Upgradeability, Reuse
 - Plan for Timely Technology Transition/ Transfer

Simultaneously Enhance Both Quality and Acceptance

Lockheed Martin Aeronautics Company

COMSAC Plan NAVAIR Comments

John Clark
25 Sep 2003



COMSAC Objectives

- Initial emphasis on validation – need to build credibility within the S&C user community
- Accelerate application in selected areas where chance of success is greatest
- Focus on both near and far term objectives
- Emphasize synergistic application of CFD, wind tunnel, PID, engineering methods, ...

Objectives (con'd.)

- Assess State of the Technology - produce report card similar to that presented by Bogue – clearly define alignment of tools and problems, examples, correlations, strengths/weaknesses, ...
- Define S&C requirements – priorities and required accuracies

COMSAC Strategy

- Calibrate and validate – build credibility
- Gather data from all participants – wind tunnel and PID; consciously gather new data from planned tests; new tests
- S&C CFD Challenges in priority areas

NAVAIR Priorities

- Low AOA S&C data base development – C_{mo} , C_n and C_l derivatives
- Increments to existing configurations
- Dynamic derivatives
- Propulsion induced / ground effects
- High lift
- Unsteady effects

Observations

- Need to standardize terminology, axis systems
- Don't jump into the deep end
- Don't oversell the technology before it's time
- If we show that the application of CFD to S&C problems is worthwhile – solutions to time, cost, ease of use issues will follow

Issues

- Data availability
- Funding / sponsors
- Limited to fixed wing configurations; what about helos, tilt rotors?

“Build it and they will come”

“Validate it and they will use it”

COMSAC Feedback – Boeing Commercial Airplanes

Very impressive display of the work that has been going on in IGA

Some overlap of interest; some very diverse
Conventional controls
Active flow control

Very interesting “difference of opinion” about level of tool required to do the S&C task. Is N-S always needed?



COMSAC Workshop, September 23-25, 2003



COMSAC Feedback – Boeing Commercial Airplanes

- Areas of concern:
 - NASA metrics are emissions, noise and mobility; it may be difficult to map COMSAC to these metrics (and show a significant payoff) Is COMSAC the enabler for realizing more advanced/diverse configurations?
 - Roadmap is based upon commercial airplane data yet the makeup of the room is predominantly military. Will this roadmap “hang together” given this mix of participants?
 - Competitive issues for military much different from commercial. Can an open, sharing environment work for them? (Assumption: the ‘black program’ model is in operation”
 - And is the assumption of a ‘black program’ achievable?



COMSAC Workshop, September 23-25, 2003



COMSAC Feedback – Boeing Commercial Airplanes

- Steps for moving forward:
 - Advocating for a plan at NASA HQ demands that we HAVE a plan!
 - Create the operating environment ('black program'?)
 - Determine the critical issues to work – focus on “doing” those things that are achievable within the next five years; determine technology needs for the longer term issues and put those in the base program
 - Check: do we have an integrated program or a collection of individual problems?
 - Lay out the plan and program structure based upon the above
 - (Deliverables, exit criteria, schedule, cost)
 - Equate exit criteria to top-level NASA metrics
 - Lobby HQ for the money or go home



COMSAC Workshop, September 23-25, 2003



REPORT DOCUMENTATION PAGE					Form Approved OMB No. 0704-0188	
The public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Department of Defense, Washington Headquarters Services, Directorate for Information Operations and Reports (0704-0188), 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302. Respondents should be aware that notwithstanding any other provision of law, no person shall be subject to any penalty for failing to comply with a collection of information if it does not display a currently valid OMB control number. PLEASE DO NOT RETURN YOUR FORM TO THE ABOVE ADDRESS.						
1. REPORT DATE (DD-MM-YYYY)		2. REPORT TYPE			3. DATES COVERED (From - To)	
01- 04 - 2004		Conference Publication				
4. TITLE AND SUBTITLE COMSAC: Computational Methods for Stability and Control				5a. CONTRACT NUMBER		
				5b. GRANT NUMBER		
				5c. PROGRAM ELEMENT NUMBER		
6. AUTHOR(S) Fremaux, C. Michael and Hall, Robert M. (Compilers)				5d. PROJECT NUMBER		
				5e. TASK NUMBER		
				5f. WORK UNIT NUMBER 23-762-45-AF		
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) NASA Langley Research Center Hampton, VA 23681-2199				8. PERFORMING ORGANIZATION REPORT NUMBER L-18378B		
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001				10. SPONSOR/MONITOR'S ACRONYM(S) NASA		
				11. SPONSOR/MONITOR'S REPORT NUMBER(S) NASA/CP-2004-213028/PT2		
12. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Category 08 Availability: NASA CASI (301) 621-0390 Distribution: Nonstandard						
13. SUPPLEMENTARY NOTES An electronic version can be found at http://techreports.larc.nasa.gov/ltrs/ or http://ntrs.nasa.gov						
14. ABSTRACT The unprecedented advances being made in computational fluid dynamic (CFD) technology have demonstrated the powerful capabilities of codes in applications to civil and military aircraft. Used in conjunction with wind-tunnel and flight investigations, many codes are now routinely used by designers in diverse applications such as aerodynamic performance predictions and propulsion integration. Typically, these codes are most reliable for attached, steady, and predominantly turbulent flows. As a result of increasing reliability and confidence in CFD, wind-tunnel testing for some new configurations has been substantially reduced in key areas, such as wing trade studies for mission performance guarantees. Interest is now growing in the application of computational methods to other critical design challenges. One of the most important disciplinary elements for civil and military aircraft is prediction of stability and control characteristics. CFD offers the potential for significantly increasing the basic understanding, prediction, and control of flow phenomena associated with requirements for satisfactory aircraft handling characteristics.						
15. SUBJECT TERMS Stability and Control; Computational Fluid Dynamics; Stability Derivatives; Aerodynamics						
16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON	
a. REPORT	b. ABSTRACT	c. THIS PAGE			STI Help Desk (email: help@sti.nasa.gov)	
U	U	U	UU	389	19b. TELEPHONE NUMBER (Include area code) (301) 621-0390	